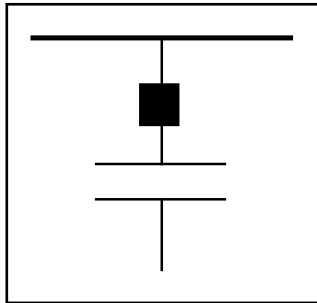
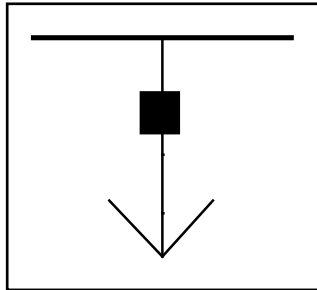
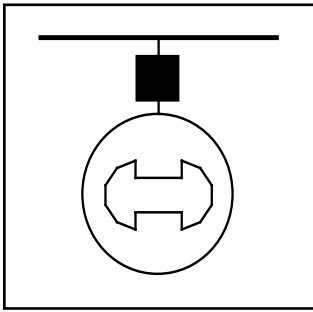
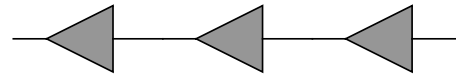




PowerWorld
Corporation



Simulator Version 16



Interactive power system
simulation, analysis and
visualization

User's Guide

December 22, 2011

2001 South First Street
Champaign, IL 61820
<http://www.powerworld.com>
support@powerworld.com
+1 (217) 384-6330

About this Manual	1
Introduction to PowerWorld Simulator.....	2
Introduction to Simulator Add-On Tools	3
What's New	5
Help.....	15
Contact Information.....	16
Windows Basics	17
PowerWorld Simulator: Getting Started	19
Edit Mode Introduction.....	20
Run Mode Introduction	21
Script Mode Introduction	22
Message Log	23
Custom and Memo Display	24
Status Bar	25
Ribbons User Interface.....	26
Quick Access Toolbar	29
Add Ons Ribbon Tab	30
Case Information Ribbon Tab	33
Draw Ribbon Tab.....	37
Clipboard Ribbon Group	38
Formatting Ribbon Group	40
Individual Insert Ribbon Group	42
Quick Insert Ribbon Group	44
Select Ribbon Group	46
Onelines Ribbon Tab.....	47
Active Ribbon Group	49
Zoom Ribbon Group.....	53
Options Ribbon Tab	54
Solution Options Quick Menu	56
Tools Ribbon Tab.....	57
Edit Mode Ribbon Group	59
Other Tools Ribbon Group	61
Run Mode Ribbon Group	63
Power Flow Tools Ribbon Group	64
Window Ribbon Tab.....	67
Application File Menu	70
Case Formats	71
Auxiliary File Format (*.aux).....	75
GE EPC File Load Options	76

PTI RAW File Load Options.....	78
Opening a Simulation Case.....	79
Opening a Oneline Diagram.....	80
Recently Opened Cases.....	82
Building a New Case	83
Building a New Oneline	84
Saving Cases	85
Saving a Oneline.....	86
Close Oneline	87
Generator Capability Curves Format (*.gcp)	88
Generator Cost Data Format (*.gcd)	89
Injection Groups Format (*.inj)	91
Interface Data Format (*.inf)	92
Sequence Data Format.....	93
Exporting Onelines in Different Graphic Formats	95
Saving Images as Jpegs	96
Saving Admittance Matrix and Jacobian Information	97
Working With GE EPC Files.....	98
Differences In File Formats.....	99
Overview of PowerWorld Simulator Project Files	103
PowerWorld Project Initialization Script	104
Associating Project Files with Simulator	105
Creating a New Project File	107
Create Project Dialog	108
Opening an Existing Project.....	110
Model Explorer	111
Model Explorer: Fields Pane.....	114
Case Information Displays.....	117
Configuring the Case Information Displays	118
Custom Field Descriptions	122
Custom Field Description Dialog.....	124
Calculated Fields.....	126
Case Information Customizations Display	128
Case Information Displays: Local Menu Options.....	129
Case Information Displays: Colors and Cell Styles	131
Case Information Displays: Using Cell Handles	132
Case Information Displays: Sorting Records	133
Case Information Displays: Finding Records	134
Saving Case Information Display Contents as HTML Tables	135
HTML Table Format Dialog	136

Case Information Toolbar.....	137
Case Information Toolbar: Copy, Paste and Send Menu	140
Case Information Toolbar: Filtering Menu.....	142
Case Information Toolbar: Geo Data View Menu	144
Case Information Toolbar: Load Auxiliary Files Menu	145
Case Information Toolbar: Save Auxiliary Files Menu	146
Case Information Toolbar: Records Menu	148
Case Information Toolbar: Set, Toggle, and Columns Menus	150
Case Information Filterbar	152
Area/Zone/Owner Filters	153
Advanced Filtering	154
Advanced Filters Dialog	155
Advanced Filters View Filter Logic Graphical Display	157
Advanced Filtering: Advanced.....	158
Advanced Filtering: Device	160
Advanced Filters Display.....	161
Custom Expressions.....	162
Custom Expressions Display	163
Custom String Expressions	164
Functions and Operators Available	165
Model Expressions	168
Find Dialog Basics.....	172
Search for Text Dialog	174
Model Conditions Display and Dialog	175
Model Filters Display and Dialog	176
Model Filters View Filter Logic Graphical Display	177
Entering a Range of Numbers.....	178
Referencing Model Expressions.....	179
Copying Simulator Data to and from Other Applications	180
Save Case Information Data	181
Key Fields	182
Required Fields.....	184
Contour Column Dialog	185
Contour Column Type	186
Grid Metrics Dialog	188
Auxiliary File Export Format Description (For both Display and Power System) ...	189
Geographic Data View.....	193
Geographic Data View Styles: Fields and Attributes.....	195
Geographic Data View Styles: General Display Options.....	199
Geographic Data View Styles.....	200

Custom Case Information Display	202
Define Fields/Strings	203
Change Field Data	205
Show Fields Primary.....	207
Show Fields Secondary	208
Custom Case Information Display Local Menu.....	209
User-Defined Case Information Displays.....	210
Case Description	212
Case Summary	214
Power Flow List	216
Quick Power Flow List	218
Outages	219
Area Display	220
Zone Display	222
Tie Lines between Areas Display	223
Tie Lines between Zones Display	225
Super Area Display	226
Bus Display	228
Bus Mismatches Display	229
Remotely Regulated Bus Display	230
Substation Records Display	232
Generator Display	233
Generator Economic Curves	235
Generator/Load Cost Models	236
Generator Cost Models Display	237
Generator Cubic Cost Display	239
Generator Piecewise Linear Cost Display	241
Load Display.....	243
Load Benefit Models Display	244
Line and Transformer Display	246
Merge Line Terminals.....	248
Long Line Voltage Profile.....	249
Multi-Section Lines Display	253
Transformer Display.....	255
Three Winding Transformer Display	257
Line Shunts Display.....	259
DC Lines Display	260
Multi-Terminal DC Record Display	261
Switched Shunt Display	262
Interface Display	264

Nomogram Display	266
Injection Group Display	267
Injection Group Dialog	269
Participation Point Records Display	271
Island Display	274
MW Transactions Display	275
Transaction Dialog	276
Owner Data Information Display	278
Owner Dialog	279
Owned Bus Records Display	280
Owned Load Records Display	281
Owned Generator Records Display	282
Owned Line Records Display	283
Jacobian Display	284
Ybus Display	285
Bus Information (Edit Mode)	286
Bus Field Information	289
Shortest Path Between Elements	291
Bus Voltage Regulating Devices Dialog	292
Substation Information (Edit Mode)	293
Substation Field Options	295
Generator Information (Edit Mode)	297
Generator Field Information	299
Generator Options: Display	301
Generator Options: Power and Voltage Control	302
Generator Options: Costs	305
Generator Options: Fault Parameters	307
Generator Options: Owners, Area, Zone, Sub	308
Generator Cost Description	309
Set Generator Participation Factors	310
Generator Reactive Power Capability Curve	312
Load Information (Edit Mode)	313
Load Options: Load Information	315
Load Options: OPF Load Dispatch	316
Load Field Information	317
Load Modeling	319
Branch Options (Edit Mode)	320
Branch Options: Display	323
Branch Options: Parameters	324
Line Shunts Information	325

Line Per Unit Impedance Calculator Dialog	326
Transmission Line Parameter Calculator	327
Branch Options: Transformer Control	337
Transformer AVR Dialog	340
Branch Options: Series Capacitor	342
Multi-Section Line Information	343
Line Field Options	345
Series Capacitor Field Options Dialog	347
Transformer Field Options Dialog	348
DC Transmission Line Options	350
DC Line Options: Line Parameters	351
DC Line Options: Rectifier Parameters	353
DC Line Options: Inverter Parameters	354
DC Line Options: Actual Flow	355
DC Line Options: OPF	356
VSC DC Line Dialog	357
Multi-Terminal DC Record Information	360
Multi-Terminal DC Bus Information	361
Multi-Terminal DC Converter Information	362
Multi-Terminal DC Line Information	364
Branch Options: Transformer Control	365
Transformers Bases and Impedances Dialog	368
Transformer Impedance Correction Table Display	369
Transformer AVR Dialog	370
Transformer Mvar Control Dialog	372
Transformer Phase Shifting Dialog	374
Transformer Field Options Dialog	376
Three Winding Transformer Information	378
Switched Shunt Information (Edit Mode)	380
Switched Shunt Field Information	383
Zone Information (Edit Mode)	385
Bus Information (Run Mode)	387
Bus Voltage Regulating Devices Dialog	390
Substation Information (Run Mode)	391
Generator Information (Run Mode)	393
Generator Information: Power and Voltage Control	395
Generator Options: Costs	398
Generator Information: OPF	400
Generator Options: Owners, Area, Zone, Sub	401
Load Information (Run Mode)	402

Branch Information Dialog (Run Mode)	404
Transformer AVR Dialog	408
Transformer Mvar Control Dialog	410
Transformer Phase Shifting Dialog	412
Transformer Impedance Correction Tables Dialog	414
Switched Shunt Information (Run Mode).....	415
Zone Information (Run Mode)	418
Labels	420
Label Manager Dialog	423
Latitude/Longitude and UTM Coordinates Conversion.....	424
Area Information	426
Area Information: Buses	428
Area Information: Gens.....	429
Area Information: Loads	430
Area Information: Area MW Control Options.....	431
Area Information: Info/Interchange	432
Area Information: Options.....	433
Area Information: Tie Lines	434
Area Information: OPF	435
Area Information: Custom	436
Area Field Options	437
Super Area Information Dialog	439
Super Area Field Information.....	442
Interface Information.....	444
Interface Element Information.....	446
Interface Field Information Dialog	447
Interface Pie Chart Information Dialog	448
Automatically Inserting Interfaces in Case.....	449
Nomogram Information Dialog	450
Injection Group Overview.....	452
Creating Injection Groups.....	453
Auto Insert Injection Groups	454
Deleting Injection Groups.....	456
Injection Group Display.....	457
Injection Group Dialog	459
Import PTI Subsystems Dialog.....	461
Participation Points Overview.....	462
Participation Point Records Display	463
Add Participation Points Dialog	466
Transaction Dialog	468

Bus View Display	470
Substation View Display	473
Labels	474
Label Manager Dialog	477
Difference Flows	478
Difference Flows: Case Types	479
Using Difference Flows.....	480
Present Topological Differences from Base Case	482
Auxiliary Files	484
Script Command Execution Dialog	486
Quick Auxiliary Files Dialog.....	487
Script Section	488
Script General Actions	489
Script Oneline Actions	507
Script Edit Mode Actions.....	509
Script Run Mode Actions	515
Script PowerFlow Related Actions	523
Script Contingency Related Actions	527
Script ATC Related Actions	531
Script Fault Related Actions	533
Script PV Related Actions	534
Script QV Related Actions.....	536
Script Topology Processing Related Actions	538
Script Transient Stability Related Actions.....	540
Data Section.....	543
Data Argument List.....	544
Data Key Fields.....	547
Data List	548
Data SubData Sections.....	549
Data ATC_Options	550
Data ATCExtraMonitor.....	551
Data ATCScenario	552
Data AuxFileExportFormatData	554
Data AuxFileExportFormatDisplay	555
Data BGCalculatedField	556
Data Bus	557
Data BusViewFormOptions	558
Data ColorMap.....	559
Data Contingency	560
Data CTG Options.....	567

Data CTGElementBlock	568
Data CustomCaseInfo	569
Data CustomColors	570
Data DataGrid	571
Data DynamicFormatting	572
Data Filter	574
Data Gen	576
Data GeoDataViewStyle	577
Data GlobalContingencyActions	579
Data HintDefValues.....	580
Data InjectionGroup	581
Data Interface	583
Data KMExportFormat	585
Data LimitSet	586
Data Load	587
Data LPVariable	588
Data ModelCondition	589
Data ModelExpression	590
Data ModelFilter	592
Data MTDCRecord	593
Data MultiSectionLine	597
Data Nomogram	598
Data NomogramInterface.....	599
Data Owner.....	600
Data PieChartGaugeStyle	602
Data PostPowerFlowActions.....	603
Data PWCaseInformation	604
Data PWFormOptions.....	605
Data PWLPOPCTGViol.....	606
Data PWLPTabRow	607
Data PWPVResultListContainer	608
Data PWQVResultListContainer	610
Data QVCurve	611
Data QVCurve_Options	612
Data SelectByCriteriaSet.....	613
Data ShapeFileExportDescription	615
Data StudyMWTransactions.....	616
Data SuperArea	617
Data TSSchedule	618
Data UserDefinedDataGrid	619

Data View	620
Script Section	621
Script Actions for Display Auxiliary Files.....	622
Data Section for Display Auxiliary File	627
Data Argument List for Display Auxiliary File.....	628
Data Key Fields for Display Auxiliary File	630
Data List for Display Auxiliary File	631
Special Data Sections.....	632
Data Subdata Sections for Display Auxiliary File.....	633
Data Common Sections	634
Data DisplayLine.....	635
Data PieChartGaugeStyle	636
Data View	637
Simulator Options	638
Power Flow Solution Options	639
Power Flow Solution: Common Options.....	640
Power Flow Solution: Advanced Options.....	643
Post Power Flow Solution Actions Dialog	647
Power Flow Solution: Island-Based AGC.....	649
Power Flow Solution: DC Options.....	651
DC Power Flow Loss Setup	653
Power Flow Solution: General.....	654
Power Flow Solution: Storage.....	655
Message Log Options	656
Environment Options	658
Online Options.....	660
Case Information Display Options	663
File Management Options.....	666
Power Flow Tools Ribbon Group	669
Solving the Power Flow	672
Area Control.....	676
Area Control Error (ACE) Chart	679
Area Load and Generation Chart.....	680
Area Losses Chart.....	681
Area MW Transactions Chart	682
Area Average Cost Chart	683
Edit Mode Overview	684
Edit Mode General Procedures.....	685
Draw Ribbon Tab.....	687
Online Display Options	688

Anchored Objects	689
Inserting and Placing Multiple Display Fields	690
Setting Default Drawing Options	691
Area Display Objects	694
Area Display Options Dialog	695
Zone Display Objects	696
Zone Display Options Dialog	697
Super Area Display Objects	698
Super Area Display Options Dialog	699
Owner Display Objects	700
Owner Display Options Dialog	701
Area Fields on Onelines	702
Zone Fields on Onelines	703
Super Area Fields on Onelines	705
Owner Fields on Onelines	706
Bus Display Objects	707
Bus Fields on Onelines	708
Old Voltage Gauges	709
Old Voltage Gauge Options Dialog	710
Substation Display Objects	711
Substation Fields on Onelines	712
Generator Display Objects	713
Generator Fields on Onelines	714
Load Display Objects	715
Load Fields on Onelines	716
Transmission Line Display Objects	717
Line Fields on Onelines	718
Circuit Breakers on Onelines	719
Line Flow Pie Charts on Onelines	720
Line Flow Gauges	721
Line Flow Gauge Options Dialog	722
Line Flow Arrows on Onelines	723
DC Transmission Line Display Objects	724
Multi-section Line Display Objects	725
Transformer Display Objects	726
Transformer Fields on Onelines	728
Three Winding Transformer Display Objects	729
Series Capacitor Display Objects	731
Series Capacitor Fields on Onelines	732
Switched Shunt Display Objects	733

Switched Shunt Fields on Onelines	734
Interface Display Objects	735
Automatically Inserting Interface Display Objects	736
Interface Fields on Onelines	737
InterArea Flow Options Dialog	738
Interface Pie Charts on Onelines	739
Loading NERC Flowgates	740
Saving NERC Flowgates	741
Injection Group Display Objects	742
Links to Onelines and Auxiliary Files	743
Document Links on Onelines	745
Background Lines on Onelines	746
Background Lines Dialog	747
Background Rectangles on Onelines	748
Background Ellipses on Onelines	749
Background Pictures on Onelines	750
Converting Background Lines	751
Converting Background Ellipses	752
Converting Background Rectangles	753
Online Fields	754
Text on Onelines	755
Memo Text	756
Generic Model Fields	757
Supplemental Data Fields on Onelines	758
Pie Charts/Gauges: Lines	759
Pie Charts/Gauges: Interfaces	761
Pie Charts/Gauges: Pie Chart/Gauge Styles	762
Pie Charts/Gauges: General Options	763
Pie Chart / Gauge Dialogs	764
Pie Chart / Gauge Style Dialog	767
Pie Chart / Gauge Style Dialog - Standard Parameters Tab	769
Pie Chart / Gauge Style Dialog - Open Parameters Tab	771
Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab	772
Pie Chart / Gauge Style Dialog - Gauge Parameters Tab	773
Pie Chart / Gauge Example	775
Palette Overview	776
Using the Insert Palettes	777
Quick Insert Ribbon Group	780
Automatically Inserting Buses	782
Automatically Inserting Transmission Lines	783

Automatically Inserting Generators	785
Automatically Inserting Loads.....	786
Automatically Inserting Switched Shunts	787
Automatically Inserting Interface Display Objects	788
Automatically Inserting Substations	789
Automatically Inserting Borders	790
Clipboard Ribbon Group	793
Select by Criteria Dialog	795
Select Ribbon Group	797
Grid/Highlight Unlinked Objects	798
Using the Online Alignment Grid	799
Setting Background Color	800
Zoom, Pan, and Find.....	801
Default Drawing Values	803
Formatting Ribbon Group	804
Format Multiple Objects	806
Paste Format Dialog.....	807
Font Properties	808
Line/Fill Properties	809
Levels/Layers Options	810
Screen Layers.....	811
Screen Layer Options	812
Format Field Properties	813
Display/Size Properties.....	814
Align Group Objects	815
Refresh Anchors	816
Clipboard Ribbon Group	817
Undo Command.....	819
Online Diagram Overview	820
Relationship Between Display Objects and the Power System Model	821
Custom Hint Values	822
Online Local Menu	824
Online Display Options Dialog	826
Display Options	827
Pie Chart/Gauge Options.....	829
Animated Flows Options.....	831
Thumbnail View.....	834
Thumb Nail Menu	835
Display Object Options.....	836
Substation Display Options.....	838

Online Animation	840
Copying Onelines to Other Programs.....	841
Display Explorer.....	842
List Unlinked Display Objects.....	844
All Display Objects	846
Supplemental Data	847
Movie Maker.....	849
Map Projections.....	850
Great Circle Distance Dialog	854
Great Circle Distance Calculation.....	855
Populate Lon,Lat with Display X,Y	856
Geography/Coordinates.....	857
Shape File Import	858
GIS Shapefile Data: Control	859
GIS Shapefile Data: Identify	860
GIS Shapefile Data: Modify.....	861
GIS Shapefile Data: Format	862
GIS Shapefile Data: Shape List	863
Shapefile Database Record Dialog.....	864
Export Online As Shapefile.....	865
Insert Measure Line	867
Delete All Measure Lines	868
Path Distances from Bus or Group	869
Closet Facilities to Point	870
Online Screen Coordinates.....	871
Online Zooming and Panning	872
Save View Level Dialog	874
Window Ribbon Tab.....	876
Online Conditional Display of Objects	879
Keyboard Short Cut Actions Dialog.....	880
Printing Online Diagrams	882
Print Options Dialog	883
Printer Setup	885
Contouring	886
Contouring Options.....	887
Contour Type	888
Contour Type Options	892
Custom Color Map	894
Functional Description of Contour Options.....	896
Dynamic Formatting Overview	900

Dynamic Formatting Dialog	901
Difference Flows	903
Difference Flows: Case Types	904
Using Difference Flows.....	905
Present Topological Differences from Base Case	907
Limit Monitoring Settings.....	909
Limit Monitoring Settings and Limit Violations Dialog.....	910
Limit Group Dialog	915
Difference Flows	918
Difference Flows: Case Types	919
Using Difference Flows.....	920
Present Topological Differences from Base Case	922
Scaling	924
Path Distances from Bus or Group	928
Shortest Path Between Elements	929
Find Circulating MW or MVar Flows Dialog	930
Find Branches that Create Islands.....	934
Overview of Facility Analysis in PowerWorld Simulator	935
Facility Analysis Dialog	936
Augmenting Path Max Flow Min Cut Algorithm.....	938
Graph Flow	939
Governor Power Flow.....	940
Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab..	941
Governor Power Flow: Options Tab.....	942
Set Generator Participation Factors	943
Set Selected Field	945
Network Cut	946
Create New Areas for Islands	948
Browse PWB File Headers.....	950
Browse Open Onelines	951
Unused Bus Numbers.....	952
Equivalents	953
Equivalents Display	954
Select The Buses	957
Appending a Case.....	959
Merging Buses.....	960
Splitting Buses.....	961
Split Bus Dialog	962
Equipment Mover	963
Potential Misplacements Dialog.....	964

Tapping Transmission Lines.....	965
Automatic Line Tap Dialog.....	966
Merge Line Terminals.....	968
Bus Renumbering Dialog	969
Bus Renumbering: Automatic Setup of Bus List Options	970
Bus Renumbering: Bus Change Options	972
Renumber Areas/Zones/Substations Dialog	973
Power Transfer Distribution Factors	974
Power Transfer Distribution Factors Dialog	975
Directions Display	979
Directions Dialog	980
Auto Insert Directions	981
Calculate MW-Distance.....	982
MW-Distance Options.....	983
Line Outage Distribution Factors (LODFs)	984
Line Outage Distribution Factors Dialog.....	986
Advanced LODF Calculation Dialog.....	989
Transmission Loading Relief Sensitivities	990
TLR Sensitivities Dialog	991
TLR Multiple Device Type	994
Generation Shift Factor Sensitivities	996
Loss Sensitivities.....	997
Flow and Voltage Sensitivities.....	1000
Line Loading Replicator	1005
Contingency Analysis: An Introduction	1008
Available Contingency Actions	1009
Contingency Analysis: Terms	1011
Contingency Analysis Power Flow Solution Options	1012
Contingency Case References.....	1013
Contingency Case References - State Information.....	1014
Contingency Case References - Defining the Reference State	1017
Contingency Case References - Reference State Solution Options	1018
Contingency Analysis: Defining Contingencies	1019
Auto Insert Contingencies	1020
Loading Contingencies from a File	1023
PSS/E Contingency Format.....	1024
PSS/E Load Throw Over Files.....	1025
Saving Contingency Records to a File.....	1026
Contingency Global Actions	1028
Contingency Blocks.....	1029

Contingency Analysis Dialog Overview	1030
Running the Contingency Analysis	1031
Other Contingency Actions	1032
Contingency Analysis Dialog: Contingencies Tab	1035
Contingency Definition Display.....	1038
Contingency Violations Display	1040
Contingency Definition Dialog	1042
Contingency Analysis Dialog: Options Tab	1045
Contingency Options: Modeling Basics	1046
Contingency Options: Generator Post-Contingency AGC	1049
Contingency Options: Bus Load Throw Over Records	1050
Contingency Options: Generator Maximum MW Response	1051
Contingency Options: Generator Line Drop and Reactive Current Compensation	1052
Contingency Options: Post-Contingency Auxiliary File.....	1053
Contingency Options: Limit Monitoring	1054
Contingency Options: Advanced Limit Monitoring.....	1055
Monitoring Exceptions	1058
Define Monitoring Exceptions Dialog	1060
Custom Monitors	1062
Contingency Options: Contingency Definitions	1064
Contingency Options: Distributed Computing.....	1065
Contingency Options: Miscellaneous	1066
Contingency Analysis: Results Tab	1067
Contingency Results: View Results By Element	1068
Contingency Results: View Results By Element - Contingencies Section	1069
Contingency Definition Display.....	1070
Contingency Options Tab: Report Writing.....	1072
Contingency Results: Summary	1074
Comparing Contingency Analysis Results.....	1075
Comparing Contingencies List Displays.....	1077
Contingency Element Dialog	1080
Contingency Element: Open Breakers	1093
Make-Up Power Sources.....	1095
Contingency Make-Up Sources Dialog	1096
Tutorial: Contingency Analysis.....	1097
Tutorial: Contingency Analysis - Page 2	1098
Tutorial: Contingency Analysis - Page 3	1101
Tutorial: Contingency Analysis - Page 4	1103
Tutorial: Contingency Analysis - Page 5	1105
Tutorial: Contingency Analysis - Page 6	1106

Tutorial: Contingency Analysis - Page 7	1108
Tutorial: Contingency Analysis - Page 8	1109
Tutorial: Contingency Analysis - Page 9	1110
Tutorial: Contingency Analysis - Page 10	1111
Tutorial: Contingency Analysis - Page 11	1113
Time Step Simulation	1114
Time Step Simulation: Quick Start	1115
Time Step Simulation Dialog.....	1117
Time Step Simulation Toolbar.....	1120
Time Step Simulation Pages	1121
Time Step Simulation: Summary	1122
Time Step Simulation: Summary Local Menu	1124
Time Step Simulation: Input.....	1125
Matrix Grids.....	1126
Time Step Simulation: Results Constraints.....	1127
Time Step Simulation: Binding Elements Dialog	1131
Time Step Simulation: Results	1132
Time Step Simulation: Custom Results Selection Dialog.....	1133
Time Step Simulation: Results Grids.....	1134
Time Step Simulation: Options	1135
Time Step Simulation: Specifying and Maintaining a List of Timepoints	1139
Time Step Simulation: New Timepoint Dialog	1140
Time Step Simulation: Change Timepoint Time Dialog	1141
Time Step Simulation: TSB Case Description	1142
Time Step Simulation: Insert/Scale Column Dialog.....	1143
Time Step Simulation: Loading Input Data	1145
Time Step Simulation: Setting up Scheduled Input Data.....	1146
Time Step Simulation: Schedule Dialog	1147
Time Step Simulation: Schedules	1150
Time Step Simulation: Schedule Subscription Dialog	1152
Time Step Simulation: Schedule Subscriptions Page.....	1154
Time Step Simulation: Application of Controller Time Delays	1156
Switched Shunt Control: Time Step Options	1158
Time Step Actions.....	1162
Transformer Control: Time Step Options	1163
Time Step Simulation: Running a Timed Simulation	1166
Time Step Simulation: Running OPF and SCOPF Simulations	1167
Time Step Simulation: Storing Input Data and Results	1170
Fault Analysis	1171
Fault Analysis Dialog	1172

Fault Analysis Bus Records	1173
Fault Analysis Generator Records	1174
Fault Analysis Line Records	1175
Mutual Impedance Records	1176
Mutual Impedance Record Dialog	1177
Fault Analysis Load Records	1178
Fault Analysis Switched Shunt Records	1179
PowerWorld Simulator PV/QV Overview	1180
PV Curves	1181
PV Curves Dialog	1182
PV Curves: Setup	1184
PV Curves Setup: Common Options	1185
PV Curves Setup: Advanced Options	1190
PV/QV Quantities to Track	1195
PV Curves: Limit Violations	1200
PV Curves: Output	1203
PV Curves: QV Setup	1206
PV Curves: PV Results	1208
PV Curves Results: Overview	1211
PV/QV Curves Results: Plot	1215
PV/QV Curves Results: Track Limits	1217
PV Curves New Plots	1218
QV Curves	1219
QV Curves Dialog	1221
QV Curves: Buses	1223
QV Curves: Options	1225
QV Curves Options: Solution	1226
QV Curves Options: Contingencies	1228
QV Curves Options: Output	1230
PV/QV Quantities to Track	1234
QV Curves: Results	1239
QV Curves Results: Listing	1240
PV/QV Curves Results: Plot	1242
PV/QV Curves Results: Track Limits	1244
PV/QV Refine Model	1245
PowerWorld Simulator Optimal Power Flow Overview	1247
OPF Objective Function	1248
OPF Equality and Inequality Constraints	1249
OPF Equality Constraints	1250
OPF Inequality Constraints	1251

Determining Set of Active Inequality Constraints.....	1253
OPF Unenforceable Constraints.....	1254
OPF Marginal Costs.....	1255
OPF Primal LP.....	1256
OPF Future Enhancements	1257
OPF Options and Results.....	1258
OPF Options: Common Options	1259
OPF Options: Constraint Options.....	1261
OPF Options: Control Options.....	1263
OPF Options: Advanced Options	1266
OPF Options: Solution Summary	1267
OPF Options: All LP Variables	1270
OPF Options: LP Basic Variables	1271
OPF Options: LP Basis Matrix.....	1272
OPF Options: Bus MW Marginal Price Details	1273
OPF Options: Bus MVAR Marginal Price Details	1274
OPF Options: Bus Marginal Controls.....	1275
OPF Options: Inverse of LP Basis.....	1276
OPF Options: Trace Solution.....	1277
OPF Area Records.....	1278
OPF Bus Records	1280
OPF Generator Records.....	1281
OPF Interface Records	1284
OPF Nomogram Records	1285
OPF Line/Transformer Records	1286
OPF DC Lines Records.....	1287
OPF Load Records	1288
OPF Super Area Records	1290
OPF Controls	1292
OPF Phase Shifter Records	1293
OPF Example - Introduction	1294
OPF Example - Marginal Costs.....	1297
OPF Example - Super Areas.....	1298
OPF Example - Enforcing Line MVA Constraints.....	1300
Security Constrained Optimal Power Flow Overview	1302
SCOPF Objective Function	1303
SCOPF Dialog	1304
SCOPF Solution Process	1305
SCOPF Results.....	1306
SCOPF Equality and Inequality Constraints	1307

SCOPF Equality Constraints	1308
SCOPF Inequality Constraints	1309
SCOPF Control	1311
SCOPF CTG Violations	1314
SCOPF LP Solution Details	1316
SCOPF All LP Variables	1317
SCOPF LP Basic Variables	1319
SCOPF LP Basis Matrix	1320
SCOPF Bus Marginal Price Details	1321
SCOPF Bus Marginal Controls	1322
SCOPF Example: Introduction.....	1323
SCOPF Example: Marginal Prices.....	1326
SCOPF Example: Unenforceable Constraints	1327
Optimal Power Flow Reserves Overview	1328
OPF Reserves Topics.....	1329
OPF Reserves Controls	1330
Generator and Load OPF Reserves Bids.....	1331
OPF Reserves Constraints.....	1332
Area and Zone OPF Reserve Requirement Curves	1333
OPF Reserves Objective Function.....	1334
OPF Reserves Results and Pricing	1335
OPF Reserves Case Information Display	1336
OPF Reserves Example.....	1337
Available Transfer Capability (ATC) Analysis	1341
Available Transfer Capability Dialog	1342
ATC Dialog: Options.....	1343
ATC Dialog Options: Common Options	1344
ATC Dialog Common Options: Transfer Result Reporting Options	1348
ATC Dialog Options: Advanced Options	1351
ATC Extra Monitors Dialog	1356
ATC Dialog: Result.....	1358
ATC Dialog: Analysis	1359
Multiple Scenario Available Transfer Capability Dialog	1361
Available Transfer Capability : Distributed Computing	1363
Multiple Scenario ATC Dialog: Scenarios.....	1364
Multiple Scenario ATC Dialog: Results	1367
Multiple Scenario ATC Analysis Results: Local Menu Options.....	1369
Multiple Scenario ATC Dialog: Combined Results.....	1370
ATC Dialog: Transfer Limiters Display	1371
ATC Analysis Methods: Solution Methods	1375

ATC Analysis Methods: Single Linear Step (SL)	1376
ATC Analysis Methods: Iterated Linear Step (IL)	1377
ATC Analysis Methods: Iterated Linear Step (IL) then Full CTG Solution	1380
Simulator Automation Server (SimAuto) Overview	1384
Installing Simulator Automation Server	1385
Including Simulator Automation Server Functions	1386
Connecting to Simulator Automation Server	1387
Passing Data to the Simulator Automation Server	1389
Getting Data from the Simulator Automation Server.....	1390
Simulator Automation Server Functions.....	1391
ChangeParameters Function	1392
ChangeParametersSingleElement Function	1393
ChangeParametersSingleElement Sample Code	1394
ChangeParametersMultipleElement Function.....	1396
ChangeParametersMultipleElement Sample Code.....	1397
ChangeParametersMultipleElementFlatInput Function	1399
ChangeParametersMultipleElementFlatInput Sample Code.....	1400
CloseCase Function	1402
CloseCase Function: Sample Code	1403
GetFieldList Function.....	1404
GetFieldList Function: Sample Code	1405
GetParametersSingleElement Function	1406
GetParametersSingleElement Function: Sample Code Borland® Delphi	1408
GetParametersSingleElement Function: Sample Code Matlab®	1410
GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel	1412
GetParametersMultipleElement Function.....	1414
GetParametersMultipleElement Sample Code Borland® Delphi	1416
GetParametersMultipleElement Function: Sample Code Matlab®	1417
GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel	1419
GetParametersMultipleElementFlatOutput Function	1421
GetParameters Function	1422
ListOfDevices Function	1423
ListOfDevices Function: Sample Code for Borland® Delphi.....	1425
ListOfDevices Function: Sample Code for Matlab®	1427
ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel	1429
ListOfDevicesAsVariantStrings Function	1431
ListOfDevicesFlatOutput Function	1432
LoadState Function	1433
LoadState Function: Sample Code.....	1434

OpenCase Function	1435
OpenCase Function: Sample Code	1436
ProcessAuxFile Function	1437
ProcessAuxFile Function: Sample Code.....	1438
RunScriptCommand Function.....	1439
RunScriptCommand Function: Sample Code	1440
SaveCase Function.....	1441
SaveCase Function: Sample Code	1442
SaveState Function	1443
SaveState Function: Sample Code.....	1444
SendToExcel Function	1445
SendToExcel Function: Sample Code.....	1446
TSGetContingencyResults Function	1447
TSGetContingencyResults Sample Code	1449
WriteAuxFile Function	1453
WriteAuxFile Function: Sample Code.....	1454
Simulator Automation Server Properties	1456
ExcelApp Property	1457
ExcelApp Property: Sample Code	1458
CurrentDir Property.....	1459
CurrentDir Property: Sample Code	1460
ProcessID Property	1461
ProcessID Property: Sample Code.....	1462
RequestBuildDate Property	1463
PowerWorld Object Variables.....	1464
Simulator Automation Server (version 9)	1465
Installing Simulator Automation Server (version 9)	1466
Including Simulator Automation Server Functions (version 9)	1467
Connecting to Simulator Automation Server (version 9)	1468
Simulator Automation Server Properties (version 9)	1470
ExcelApp Property (version 9).....	1471
Simulator Automation Server Functions (version 9)	1472
ChangeParameters Function (version 9)	1473
CloseCase Function (version 9)	1474
GetParameters Function (version 9)	1475
ListOfDevices Function (version 9)	1476
LoadContingencies Function (version 9)	1477
OpenCase Function (version 9)	1478
ProcessAuxFile Function (version 9)	1479
RunScriptCommand Function (version 9).....	1480

SaveCase Function (version 9).....	1481
SendToExcel Function (version 9)	1482
WriteAuxFile Function (version 9)	1483
PowerWorld Object Variables (Version 9)	1484
Integrated Topology Processing Overview	1486
Integrated Topology Processing: Full-Topology Model.....	1488
Integrated Topology Processing: Superbuses	1490
Integrated Topology Processing: Subnets.....	1491
Integrated Topology Processing: Consolidation.....	1492
Integrated Topology Processing Consolidation: Primary Bus Priority Scheme	1493
Integrated Topology Processing Consolidation: Illustration of Process.....	1494
Integrated Topology Processing: Consolidated Representation	1495
Integrated Topology Processing: Consolidated Superbus View.....	1496
Integrated Topology Processing Dialog	1498
Integrated Topology Processing: Power Flow Using Topology Processing	1500
Integrated Topology Processing: Ill-Conditioned Jacobian	1501
Integrated Topology Processing: Contingency Analysis Using Topology Processing	1502
Contingency Element: Open Breakers	1504
Integrated Topology Processing: Incremental Topology Processing.....	1506
Integrated Topology Processing: Topology Processing in Applications	1507
Integrated Topology Processing: Saving the Planning Case	1508
Integrated Topology Processing: Planning Cases	1509
Integrated Topology Processing: Saving the Planning Case	1510
Integrated Topology Processing: Primary Bus Mapping	1511
Integrated Topology Processing: Breaker-to-Device Contingency Conversion....	1513
Device Derived Status	1514
Transient Stability Overview	1518
Transient Stability Overview: Generator Models	1519
Transient Stability Overview: Wind Modeling	1523
Transient Stability Overview: Loads	1526
Transient Stability Overview : DC Transmission Lines.....	1527
Transient Contour Toolbar.....	1528
Transient Stability Numerical Integration.....	1530
Transient Stability Overview: PlayIn.....	1532
Transient Stability: Model Data Management.....	1534
Transient Stability Data: Object Dialogs	1535
Transient Stability Case Info Menu.....	1538
Transient Stability Model Explorer	1541
Transient Stability Data: Block Diagrams	1544
Transient Stability Data from External files	1545

Transient Stability Analysis Dialog.....	1547
Transient Stability Analysis: Simulation	1550
Transient Stability Contingency Element Dialog.....	1552
Transient Stability Analysis: Options.....	1555
Transient Stability Analysis Options: General	1556
Transient Stability Analysis Options: Power System Model	1557
Transient Stability Dialog Options: Result Options.....	1559
Transient Stability Dialog Options: Generic Limit Monitors	1562
Transient Stability Dialog Options: Distributed Computing	1564
Transient Stability Analysis: Results Storage.....	1565
Transient Stability: Results Storage: Storage to RAM	1567
Transient Stability: Results Storage: Storage to Hard Drive	1571
Transient Stability Analysis: Plots.....	1573
Transient Stability Analysis Plots: Plot Designer	1577
Transient Stability Analysis Plot Designer: Plot.....	1582
Transient Stability Analysis Plot Designer: Title Block	1584
Transient Stability Analysis Plot Designer: Chart	1586
Transient Stability Analysis Plot Designer: Horizontal Axis	1588
Transient Stability Analysis Plot Designer: Vertical Axis	1590
Transient Stability Analysis Plot Designer: Plot Series List.....	1592
Transient Stability Analysis Plot Designer: Plot Series	1593
Transient Stability Dialog: User Interaction with Plots	1595
Transient Stability Transient Limit Monitors	1597
Transient Stability : Defining Transient Limit Monitors	1598
Transient Stability : Transient Limit Monitors Violations	1601
Transient Stability Analysis: Results	1603
Transient Stability Results: Time Values from RAM.....	1605
Transient Stability Analysis: States/Manual Control	1606
Transient Stability Analysis: Validation	1608
Transient Stability Dialog: SMIB Eigenvalues (Single Machine Infinite Bus)	1611
Transient Stability: Running Multiple Contingencies.....	1613
Distributed Computing Add-Ons.....	1616
Tutorial: Creating a New Case Page 1 of 13	1618
Tutorial: Inserting a Bus Page 2 of 13	1619
Tutorial: Inserting a Generator Page 3 of 13	1621
Tutorial: Saving the Case Page 4 of 13	1623
Tutorial: Entering a Second Bus with Load Page 5 of 13	1624
Tutorial: Inserting a Transmission Line Page 6 of 13	1627
Tutorial: Inserting a Line Flow Pie Chart on a Transmission Line Page 7 of 13.....	1629
Tutorial: Inserting Circuit Breakers Page 8 of 13.....	1631

Tutorial: Inserting a Transformer Page 9 of 13.....	1633
Tutorial: Inserting a Switched Shunt Page 10 of 13	1636
Tutorial: Inserting Text, Bus and Line Fields Page 11 of 13	1638
Tutorial: Solving the Case Page 12 of 13.....	1642
Tutorial: Adding a New Area Page 13 of 13.....	1644
Starting with an Existing Case Page 1 of 15.....	1647
Tutorial: Loading an Existing Power Flow File Page 2 of 15	1648
Tutorial: Case Information Displays Page 3 of 15	1649
Tutorial: Solving the Case Page 4 of 15	1650
Tutorial: Building Onelines Page 5 of 15	1651
Tutorial: Entering a Bus Page 6 of 15	1652
Tutorial: Automatic Line Insertion Page 7 of 15.....	1653
Panning and Zooming Page 8 of 15	1654
Tutorial: Adding Background Page 9 of 15	1655
Tutorial: Simulating the Case Page 10 of 15.....	1656
Tutorial: Run-time Object Dialogs Page 11 of 15	1657
Online Local Menu Page 12 of 15.....	1658
Tutorial: Area Page 13 of 15	1659
Limit Violations Page 14 of 15	1660
Other Case Information Displays Page 15 of 15.....	1661
Tutorial: Solving an OPF Page 1 of 6.....	1662
Tutorial: OPF Three Bus Example Page 2 of 6.....	1663
Tutorial: OPF Line Limit Enforcement Page 3 of 6	1664
Tutorial: OPF LMP Explanation Page 4 of 6.....	1665
Tutorial: OPF Marginal Cost of Enforcing Constraints Page 5 of 6.....	1666
Tutorial: OPF Unenforceable Constraints Page 6 of 6	1667
Tutorial: Contingency Analysis	1669

About this Manual

PowerWorld Simulator includes comprehensive, context-sensitive on-line help. Any page of this manual can be printed for your own off-line reference. Occasionally updates to the on-line help are posted in patches on the PowerWorld website.

Introduction to PowerWorld Simulator

PowerWorld® Simulator (Simulator) is a power system simulation package designed from the ground up to be user-friendly and highly interactive. Simulator has the power for serious engineering analysis, but it is also so interactive and graphical that it can be used to explain power system operations to non-technical audiences. With [Version 16](#) we've made Simulator more powerful, more visual, and easier to use.

Simulator consists of a number of integrated products. At its core is a comprehensive, robust Power Flow Solution engine capable of efficiently solving systems of up to 100,000 buses. This makes Simulator quite useful as a stand-alone power flow analysis package. Unlike other commercially available power flow packages, however, Simulator allows the user to visualize the system through the use of full-color [animated online diagrams](#) complete with [zooming and panning](#) capability. System models can be either modified on the fly or built from scratch using Simulator's full-featured graphical case editor. Transmission lines can be switched in (or out) of service, new transmission or generation can be added, and new transactions can be established, all with a few mouse clicks. Simulator's extensive use of graphics and animation greatly increases the user's understanding of system characteristics, problems, and constraints, as well as of how to remedy them.

The base package of Simulator is capable of solving power systems comprised of up to 100,000 buses. The base package also contains all the tools necessary to perform integrated economic dispatch, area transaction economic analysis, power transfer distribution factor (PTDF) computation, short circuit analysis, and contingency analysis. All of the above features and tools are easily accessible through a consistent and colorful visual interface. These features are so well integrated that you will be up and running within minutes of installation.

In addition to the features of the base Simulator package, various add-on tools are available, including the new transient stability feature. Please see [Introduction to Simulator Add-On Tools](#) for more information.

Introduction to Simulator Add-On Tools

In addition to the features of the base Simulator package, various add-on tools are available. A brief introduction to each follows:

Voltage Adequacy and Stability Tool (PVQV)

The purpose of the PVQV add-on is to allow the user to analyze the voltage stability characteristics of a system. After the PVQV simulation is complete, the user can graph various system parameters. For more information, see the [PVQV Overview](#).

Optimal Power Flow Tool (OPF)

The purpose of an OPF is to minimize an [objective \(or cost\) function](#). In Simulator OPF the Linear Programming OPF algorithm (LP OPF) determines the optimal solution by iterating between solving a standard power flow and solving a linear program to change the system controls thereby removing any limit violations. For more information, see the [OPF Overview](#).

Security Constrained Optimal Power Flow Tool (SCOPF)

The OPF tool minimizes an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. The SCOPF tool takes it one step further by considering contingencies that may arise during system operation and ensuring that in addition to minimizing the objective function, no unmanageable contingency violations occur. For more information, see the [SCOPF Overview](#).

Optimal Power Flow Reserves (OPFR)

Simulator OPF Reserves is the tool used to simulate Ancillary Services Reserve Markets. For more information, see the [Optimal Power Flow Reserves Overview](#).

Available Transfer Capability Analysis Tool (ATC)

ATC analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. For more information, see the [ATC Analysis Overview](#).

PowerWorld Simulator Automation Server (SimAuto)

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on SimAuto, see the [SimAuto Overview](#).

Integrated Topology Processing

Integrated Topology Processing allows you to solve EMS-type, full-topology models in a numerically robust and transparent manner. Integrated Topology Processing extends Simulator applications traditionally used by planners so they can operate in an operations real-time environment. For more information, see the [Integrated Topology Processing Overview](#).

Transient Stability (TS)

Transient Stability analysis allows the analysis of system dynamic response to a fault on the system. For more information, see the [Transient Stability Overview](#).

Distributed Computing

Distributed computing allows the use of multiple computers to speed up analysis by splitting the work among a user-specified list of computers. For more information, see the [Distributed Computing Add-Ons](#) topic.

What's New

Web links to previous version information: [Version 15](#) [Version 14](#) [Version 13](#) [Version 12](#) [Version 11](#) [Version 10](#) [Version 9](#) [Version 8](#) [Version 7](#) [Version 6](#)

Simulator Version 16 contains a number of major new features and several smaller enhancements designed to improve the performance and convenience of the package.

- New Add-Ons
 - [Distributed Transient Stability](#)
- Auxiliary Files and Display Auxiliary Files
 - The Selected field can now be used with oneline display objects in [Display Auxiliary Files](#).
- Auxiliary File SCRIPT and SimAuto Changes
 - All Submodes such as Contingency, PowerFlow, ATC, etc. have been made obsolete in the scripting language. The Run mode and Edit modes still exist, but otherwise moving between submodes is no longer necessary.
 - New and modified script commands
 - Added new script command [AutoInsertTieLineTransactions](#);
 - Added new script command [InjectionGroupsAutoInsert](#);
 - Added new script command [SendToExcel\(ObjectType, \[FieldList\], Filter, UseColumnHeaders, "WorkBookName", "WorkSheetName"\);](#)
 - Added new script command [DetermineBranchesThatCreateIslands\(BranchesToProcess, StoreBuses, "FileName", SetSelectedOnLines\);](#)
 - Added new script command [StopAuxFile](#);
 - Added new script command [SetGenPMaxFromReactiveCapabilityCurve\("filtername"\);](#)
 - Added new script command [SetScheduledVoltageForABus\(\[bus identifier\], voltage\);](#)
 - Added [Transient Stability script commands](#)
 - Modified the editing of data in a case information display (or through an AUX file) so that the string "&ModelExpressionName:8:4" can be entered. A string which starts with the character & will be treated as though the value of the named model expression is being entered in the field. The format of the string will be determined by the last two integer values representing the total number of digits and decimal places.
 - Added a new file type to the [SaveData\(\) script command](#) called "CSVColHeader". When using this file type, the file will be written with the first row showing column headers you would see in a case information display. The file type "CSV" will always show variable names. Similar feature was added for SaveDataWithExtra() script command.
 - Modified the [Move\(\) script command](#) to permit you to move a MULTISECTIONLINE. The format is the same as for moving a regular branch except that the object name should be MULTISECTIONLINE.
 - Added optional parameter to [SaveCase script command](#) for use when saving an EPC file. New format is SaveCase("filename", SaveFileType, [PostCTGAGC, UseAreaZone]); PostCTGAGC by default is set to NO. Set to YES to set the base load flag in the EPC file based on the Post-Contingency Prevent AGC Response setting. If preventing post-contingency AGC, the base load flag is set to 1. If not preventing post-contingency AGC or this option is set to NO, the base load flag is set to 0. UseAreaZone by default is set to NO. Set to YES to limit the entries in the EPC file based on the area/zone/owner filter.
 - Added support for Display Auxiliary File (AXD) script commands to allow manipulation of oneline diagram objects
 - The following [general commands](#) can be used: ExitProgram, LoadScript, LoadData, SelectAll, UnSelectAll, SaveData, SaveDataWithExtra, Delete, WriteToTextFile, SetData, CreateData, DeleteFile, RenameFile, CopyFile, SetCurrentDirectory, SaveObjectFields, SendToExcel, and StopAuxFile.

- The following are new [online diagram specific script commands](#)
 - AutoInsertBorders
 - AutoInsertBuses(LocationSource, MapProjection, AutoInsertBranches, InsertIfNotAlreadyShown, "FileName", FileCoordinates);
 - AutoInsertLoads(MinkV, InsertTextFields, InsertEquivObjects);
 - AutoInsertGens(MinkV, InsertTextFields);
 - AutoInsertSwitchedShunts(MinkV, InsertTextFields);
 - AutoInsertLines(MinkV, InsertTextFields, InsertEquivObjects, InsertZBRPieCharts, InsertMSLines, ZBRImpedance, NoStubsZBRs, SingleCBZBRs);
 - AutoInsertLineFlowObjects(MinkV, InsertOnlyIfNotAlreadyShown, LineLocation, Size, FieldDigits, FieldDecimals, TextPosition, ShowMW, ShowMvar, ShowMVA, ShowUnits, ShowComplex);
 - AutoInsertSubstations(Location Source, Map Projection, AutoInsertBranches, InsertIfNotAlreadyShown, "FileName", FileCoordinates);
 - AutoInsertLineFlowPieCharts(MinkV, InsertOnlyIfNotAlreadyShown, InsertMSLines, Size);
 - AutoInsertInterfaces(InsertPieCharts, PieChartSize);
 - ResetStubLocations(ZBRImpedance, NoStubsZBRs);
- Labels now supported in SimAuto functions ChangeParametersSingleElement, GetParametersSingleElement, and GetParametersMultipleElement.
- Available Transfer Capability (ATC) Enhancements
 - Optimized the speed of the ATC calculations. Overall speed increase from the initial release of Simulator 15 is now between 2 and 6 times for the iterated solution methods and between 4 and 10 times for the Single Linear Step solution methods.
 - Added a new option to [linearize the calculation of contingency makeup power](#). This will precalculate the impact of makeup power on line flows at the start of the analysis and then the total effect of makeup power will be determined by multiplying the effect on a line by the Total amount of Makeup power needed. This calculation will be slightly faster than previous calculation of makeup power, but it will not take into account the fact larger amounts of makeup power may cause particular generators to hit min or max limits.
 - Removed the option Apply PTDF Cutoff for Limiting Elements with Contingencies.
 - Added a new option for the ATC calculation when performing the calculation on multiple scenarios. The option will allow you to specify that [monitoring be done only on branches that are defined as part of the scenarios](#).
 - Added option to select the [Field to Show](#) on the multiple scenarios results summary table.
 - Modified the [highlighting used and Iteratively Found string](#) for better understanding of the results when using iterated methods.
 - Removed the Max Iterations option when using the iterated methods. This is no longer necessary as oscillation prevention is handled internally.
- Bus View and Substation View
 - Now storing option settings for [displaying MW branch, Mvar branch, and MW interface flows in absolute value](#) in the registry. The values stored in the registry will be used as the defaults for these settings when online options don't exist for the Bus View or Substation View with the power flow case (PWB).
 - [Default drawing values](#) for generator rotor shape and fill will be used with the Bus View as the default shape. If a fuel type for a generator is specified, the rotor shape will depend on that, but the default drawing values will be used if no fuel type is specified.
- Case Information Displays
 - Added ability to define a [Quick Filter](#) on a case information display. This gives the ability to define an advanced filter for temporary use on a case information display and does not require you to name the filter.
 - Added ability to use other object type filters with the GenVarLim object.

- Add ability to define [Custom String Expressions](#) which represent the concatenation of strings from the columns of an object. Included in this is a new function available in the expression string with format "Str(x1, minlength, decimals)" where x1 represents a numeric variable, minlength represents the minimum number of characters for the result, and decimals represents the number of characters to the right of the decimal point.
- When choosing [Send to Excel](#), Simulator will now programmatically set Excel to NOT "treat consecutive limiters as one". It also ensures that some "Other" character is not being used as a delimiter so that only TAB characters are used.
- Modified the editing of data in a [case information displays](#) (or through an AUX file) so that the string "&ModelExpressionName:8:4" can be entered. A string which starts with the character & will be treated as though the value of the named model expression is being entered in the field. The format of the string will be determined by the last two integer values representing the total number of digits and decimal places.
- The Transformer Configuration Field is now enterable and toggleable on a case information display.
- Modified [case information displays](#) show that when showing a value such as -0.01 which is truncated to fewer decimal places, the resulting string will still include the negative sign and show "-0.0" for example instead of "0.0".
- In a [Custom String Expression](#), when using the Str(Value, Digits, Decimals) function, if the Decimals parameter is specified as negative, then it will truncate all trailing zeros beyond the decimal.
- Added fields for Owner records showing the Number of Branches by Type (Transformer, Line, SeriesCap, Breaker, Disconnect), and the number of switched shunts.
- Added fields for injection groups that list Area/Zone/Owner of all of the devices in the injection group. Fields are available for listing either the number or name of the associated Area/Zone/Owner.
- When sending [Custom Case Information](#) to Excel, only fields that are text will be marked as text. Previously, the formatting was based on columns being identified as text based fields instead of identifying each individual cell as to whether or not it is text.
- Added the ability with [case information displays](#) to show the Super Area in which a generators, loads, switched shunts, and branches are contained.
- Amps field is now available with loads.
- With [buses](#) and [substation](#) records, added the ability to specify latitude/longitude information using the four parameters of UTM longitude zone, UTM Easting, UTM Northing and specification of either north or south hemisphere.
- Contingency Analysis Enhancements
 - Two different types of fields are now available with [contingencies](#) to list the Areas/Zones/Owners associated with the elements of the contingency. Fields exist for listing either the number or name of the associated Areas/Zones/Owners. Fields that are labeled with "recurse" will look inside injection groups, interfaces, and contingency blocks for identifying the associated Areas/Zones/Owners. Fields without "recurse" will not look inside injection groups, interfaces, or contingency blocks.
 - Added option to use labels in contingency action descriptions. This includes actions shown in the [Contingency Definition list](#) and [What Actually Occurred](#). This can be useful when used with a full-topology EMS model.
 - When listing [contingency results by element](#) (bus, branch, interface, nomogram interface, or custom monitor), only those elements which actually have a violation are now listed
 - Specific generators, loads, and switched shunts can now be specified in addition to buses when using [Move, Set To, and Change By actions](#).
 - Added an optional "inclusion filter" with each [Contingency Global Actions Elements](#). The global action will only be used with a particular contingency only if the contingency meets the inclusion filter.

- Added new [Custom Monitoring](#) to the contingency analysis tool. This allows you to setup monitoring of a field of a specific object (or all objects of a specific type) and report these fields values as limit violations in the contingency analysis. Appropriate advanced filters can also be specified so that violations are only reported if specified pre-contingency or post-contingency conditions are met.
- ["Close with Breakers"](#) action is now available. This will attempt to identify breakers that can be closed in order to energize a device rather than changing the status of the device itself.
- **Fault Analysis**
 - Added total fault impedance field to [Bus](#), to store the Thevenin R and X needed to calculate the total fault X/R ratio.
 - Added ability to define a list of [Fault Definitions](#) (similar to a list of contingency definitions). These faults can then be processed as a batch and the results of Fault Current Magnitude and Angle, and Thevenin impedance at the fault can be automatically saved for each fault.
- **File Formats**
 - When using the [Present Topological Differences from Base Case](#) tool, added the ability to save the removed elements in the EPC file format using status values of -4.
 - Modified the writing of GE EPC file so that Multi-Terminal DC converters will write out the "aloss" parameter in a manner consistent with how the EPCL file mtTAP.p expects to ensure the EPCL operates appropriate for WECC data files.
 - When reading from an EPC file, added the reading of the area and zone designation of a GE branch and transformer.
 - Added a log warning message when appending an EPC file which causes a transformer branch to be converted to a normal branch.
 - When loading in EPC files and the latitude and longitude values are both zero, both entries will be ignored as invalid.
 - When appending an EPC file with a scheduled voltage for a bus, the voltage setpoints for switched shunts and generators that are regulating this bus are now properly updated.
 - Modified how appending an existing three-winding transformer is handled if a new three-winding transformer is entered which uses a bus number which is already used as a star bus by another transformer. This is not allowed and a new bus number will automatically be chosen.
 - When appending to an existing case using an EPC file, any new branches that are created are now flagged as having just been added. This flag is then used as part of the pre-processing feature in the power flow solution to automatically estimate appropriate voltage magnitudes and angles for any existing or new buses that become newly energized.
 - Modified the writing of PTI RAW files so that switched shunt records always write out at least one block. If none exists, a single block with 1 step and size equal to the present MVar output is written.
 - When appending using the RAW file format, we now populate a field to say whether an object is new or modified while reading.
 - Added an option when [saving a RAW file](#) to append object labels as comments to the end of data records.
 - When appending to an existing case using a RAW file, any new branches that are created are now flagged as having just been added. This flag is then used as part of the pre-processing feature in the power flow solution to automatically estimate appropriate voltage magnitudes and angles for any existing or new buses that become newly energized.
 - When loading the hdbexport file, modified to automatically add two labels to each transmission line. One label starts with the "from" substation name and the other label starts with the "to" substation name.
 - Modified to add additional bus labels when reading the MEAS records in the Areva CSV file.
 - At the end of reading in an [hdbexport CSV file](#), if any CBTyp values were encountered that were not recognized a dialog box will appear prompting the user to designate a Branch

- Device Type for each CBTyp. In addition options can be specified in the Simulator Options dialog under File Management that specifies a mapping from CBTyp to Branch Device Type.
- Added options when saving a case in a PWB file to force [saving the case with comments](#).
 - Added [file menu choices to open a oneline](#) using particular linking (by Number, by Name_kV, or by Label) and then allow storing the default choice so that onelines can always default to be opened by Label or Name_kV if desired. Done to work better with full-topology cases where labels are consistent.
 - Added support for EPC version 18.
 - General
 - Modified how Simulator handles setting its Windows Operating System Priority so that the user can change this. The only exception is that while animating, Simulator always sets its Priority to Low to ensure that the animation does not interfere with other programs.
 - Various dialog messages now distinguish between the various [branch device types](#), i.e. breaker, disconnect, etc., instead of just identifying as a branch.
 - Added a [graphical display to visualize the logic diagram for a Model Filter](#). This shows the logic diagram including AND and OR gates with the associated Model Filters and Model Conditions which are combined to give the Model Filter.
 - Added a [graphical display to visualize the logic diagram for an Advanced Filter](#). This shows the logic diagram including AND and OR gates with the associated advanced filters and conditions which are combined to give the Advanced Filter.
 - Geography Tab in [Bus Dialog](#) now converts from Lat/Lon to UTM and vice versa.
 - Added [Voltage Source Converter \(VSC\) DC Lines](#).
 - When defining [Model Filters](#), a NOT operator is available that can be applied to each Model Condition or Model Filter that is part of the filter.
 - Integrated Topology Processing
 - Added a new field for object types Branch, Gen, Load, Shunt, and DCLine which shows the [Derived Status](#). For full topology power system models, this field will show whether the breakers adjacent to a device are OPEN or CLOSED. If the device is energized, the field will always show CLOSED. If the device's actual Status is OPEN, the field will always show OPEN. If the device's actual status is CLOSED, but the device is not energized, the software will determine if any adjacent Breaker objects are CLOSED in which case the derived status will show CLOSED - otherwise it will show OPEN.
 - [Branches that are part of an interface will never be consolidated](#). In order to improve consolidation, if an interface contains a branch that can be consolidated, i.e. breaker, disconnect, etc., but it is in series with a device that cannot be consolidated, i.e. branch, load, gen, etc., the interface will be automatically modified to monitor the device in series instead.
 - Added optional parameter, AddCommentsForObjectLabels, to the [SaveConsolidatedCase](#) script command. This parameter is set to NO by default. If set to YES and saving to a RAW file, object labels will be added to the end of data records. The new format of the script command is SaveConsolidatedCase(filename,filetype,[BusFormat, TruncateCTGLabels, AddCommentsForObjectLabels]).
 - On several dialogs which show results of calculations for all buses in the system, an option was added to the Buses tab saying "Only show the primary bus for each superbus". This is done to reduce the number of duplicate results shown when using a full-topology model. Dialogs effected by this changed include the [Limit Monitoring dialog](#), [TLR/GSF Sensitivities dialog](#), [Loss Sensitivities](#), and [Flows and Voltages Sensitivities](#).
 - Added new [Branch Device Types](#) for Fuse, Ground Disconnect, and Load Break Disconnect. The load break disconnect is treated as a special switching device similar to the Breaker, so special actions such as "Open with Breakers" will automatically open load break disconnects.
 - Added option to [Close Breakers to Energize Switched Shunts](#). When using this option, breakers will be closed automatically if a switched shunt is on either Discrete or Continuous

- control and is required to meet the regulated voltage. This option is applied any time that the power flow is solved.
- When using the [Economic Merit Order Dispatch](#) method with the PV tool and a generator is needed in the dispatch, breakers will be closed automatically to energize the generator if necessary.
- Added script command [CloseWithBreakers](#).
- Online Diagrams
 - When using a contour with a dynamic interpretation, if all the values being contoured had the same value the contour image could appear random. This was remedied by enforcing a minimum tolerance on the separation between minimum, nominal, and maximum values of the color map.
 - Added a [Simulator Online option](#) that alter how Simulator responds when click on a circuit breaker symbol for a generator display object. Normally the generator's status is set to closed and the MW value is reset to its previous value. The new option allows the value to be set to its minimum MW output instead.
 - When changing to a Saved View which specifies that a contour be shown, we now automatically refresh the contour image
 - Added ability to [export a online](#) diagram as a GIF image.
 - Added ability to [export a online](#) diagram as an animated GIF image (animated the arrows and generator rotors).
 - When opening a online diagram using the AXD format, if no Saved View named THIS_VIEW_IS_APPLIED_TO_THE_CURRENT_DISPLAY is specified, then a Show Full will automatically occur for the online.
 - When clicking on a online diagram, substation online objects have been modified to give preference for clicking on the substation. If another object overlaps the substation, Simulator will still register the click on the substation now.
 - For dynamic formatting purposes, Model Fields will behave like the type of field that they are showing. Thus, a model field showing Area information will behave as though it is an Area Field.
 - Modified [Line Flow Arrow objects](#) so that the text better aligns with the arrow.
 - Added a new [Interface Flow Arrow Object](#) which behaves very similar to the Line Flow Object show an arrow to represent the MW flow direction and then shown text for MW value and Mvar value with a sign relative to the MW flow direction.
 - Previously an option existed in Simulator which allowed you to specify whether to show transformer symbols using either Coil or Circles. We now also have a online-specific option which allows this option to be different for different onelines.
 - Added [options](#) to automatically load a Display Auxiliary File (AXD) with each online that is opened. One option stores the settings in the windows registry so this applies to all onelines opened on the computer. Another option stores the settings in the case (PWB) so that it only applies to onelines opened with the specific case.
 - Added an [option](#) stored with the case to specify the maximum number of onelines that can be open simultaneously (this option defaults to 10). If this maximum threshold is exceeded, Simulator will automatically close the online whose most recent access is oldest. By default a online can be closed in this manner, but an additional online-specific option has been added that can be changed to prevent this closure. This feature has been added for users who make extensive use of online links or substations display object links which can make opening dozens of onelines very easy. Opening dozens of onelines can slow down the program substantially. If you have a main online from which other onelines are automatically opened, you may want to prevent the main online from closing.
 - Added [transparency with contouring](#). Transparency can be defined with either the No Data Color or with a [custom color map](#).
- Power Flow Solution

- Modified slightly when generator limits are checked. Previously they would be checked "immediately" (i.e. inside the inner power flow loop) if the mismatch fell below 0.02 per unit (2 MW or 2 Mvar normally). This could lead to some inconsistent results if in one situation the mismatch went from 2.001 to near zero, and in another it went from 1.999 to near zero. Simulator will now only check generator limits inside the inner power flow loop if the user chooses the "check immediately" solution option.
- Modified the algorithm which [automatically chooses a slack bus](#) when a new island is created. Previously, if possible, it would always choose a bus which had generation that regulated its own terminal bus. Now it is biased to do this, but if another bus with generation that does not regulate its terminal bus is available and this other bus has a maximum MW output that is 5 times higher, it will be chosen.
- When pre-processing the voltage profile of a solution before solving, Simulator will now look at groupings of buses connected by very low impedances lines. If a bus in a grouping of energized buses has a zero voltage while other buses in the group do not, the zero voltage will be changed to the first non-zero voltage found in the grouping. This provides a much more reliable solution.
- Added a new field, Enforce Current Limit, with multi-terminal DC line converters. When set to NO, the current limits for the converters will not be enforced. The default setting for this field is YES. When loading multi-terminal DC lines from an EPC file, this option will be set to NO.
- Multiple switched shunts can now be on control at the same bus. Coordination of control for shunts that regulate the same bus (or same bus due to ZBR grouping) is handled through a [Var Sharing Parameter](#) that can be specified with each shunt.
- PV and QV Curve Enhancements
 - When choosing to report limit violations in the PV Curve Tool, if no limit violations are found then the minimum voltage is reported instead.
 - In the [PV curve tool](#), added the ability to estimate at what transfer level a voltage becomes inadequate. Estimation is done by linearly interpolating between the two transfer levels where it goes from adequate to inadequate.
 - When exporting QV results to a CSV file, all results are now sorted by the voltage setpoint.
 - Added the ability to plot multiple QV curves on the same plot.
 - The more flexible plotting environment available in Transient Stability has been integrated into the PV Curve tool. This is available under "[New Plots](#)" on the PV Curve dialog. Note that this is available in addition to the previous plotting tool.
 - PV Curve Tool state archiving is now available when using Integrated Topology Processing.
 - Nominal Shift field is now included in the PV output file. Previously, only the Export and Import fields were included sometimes making it difficult to match up results between a scenario and the base case.
 - Optimized the solution speed of the PV and QV curves tools. The QV curve especially should run much more quickly, especially on cases smaller than a few thousand buses.
 - Added a new YES/NO optional parameter to [PVWriteResultsAndOptions](#)("filename", AppendFile). Specifying NO will cause it to overwrite the existing file. Specifying YES will cause it to append to the file. If not specified, then YES is assumed.
 - Attempt to speed up the process of determining how much additional var support is necessary for the power flow solution to converge when a contingency will not solve in the base case. Now look for the solvable point by tracing the QV curve from the present base case voltage up to the maximum voltage instead of tracing from the present base case voltage down to the minimum voltage.
 - Added Economic Merit Order Dispatch option for use when dispatching generators. This option allows specifying how source or sink generators are dispatched at various transfer levels. Ideally, generators will maintain their output within an economic range. Generators will be turned on or off as needed to maintain all online generators within their specified

economic range. Generators that are not needed to provide MW output will also not provide Mvar output.

- Security Constrained OPF (SCOPF)
 - When using the DC-Approximation, modified the SCOPF dialog so that less messaging is sent to the SCOPF dialog regarding the progress of the SCOPF solution. All the messaging to status edit boxes on dialog was greatly slowing down the solution process.
- Sensitivity Calculations
 - Greatly increased the speed of calculating multiple element TLR sensitivities.
 - Modified internal numerical routines to make all sensitivity calculations faster.
 - Added [interface fields](#) for showing the LODF and the Post-CTG MW flow (after Line outage) values similar to those presently available for a branch.
 - Previously it was only possible to open one instance of the [LODF dialog](#) at a time. This was because only one LODF result was stored so performing the calculation on one dialog changed the results on other open dialogs even if the chosen branch did not change on the other dialogs. This caused confusion and thus resulted in us restricting to one LODF dialog. This has now been modified to allow two LODF dialogs to be open simultaneously, but when doing this Simulator forces one dialog to show a "Single LODF" calculation, while the second dialog shows the "LODF Matrix". This is less confusing because the "LODF matrix" results are stored separately and thus can be displayed separately.
 - Same modification done for [PTDF dialog](#) by restricting the first dialog to "Single" Directions, while the second dialog shows "Multiple" directions.
 - Same modification done for [TLR/GSF dialog](#) by restricting the first dialog to the "Line/XFMR or Interface" device type, while the second dialog shows "Multiple Elements" device type.
 - Added two new Device Type/ Flow Type combinations on the [Flows and Voltages Dialog](#), under Single Meter, Multiplier Transfers. Bus / Mvar allows you to see the sensitivity of the total generator Mvar injection at a bus due to additional injections of MW or Mvar at all buses in the system. Gen / Mvar allows you to see the sensitivity of the Mvar output of a single generator to additional injections of MW or Mvar at all buses in the system.
 - Added a new tab on the Flows and Voltages tab called [Multiple Meters, Single Control](#) Change. This allows the calculation of the sensitivity of various quantities to a change in a control setpoint. Control setpoints include generator voltage setpoint, transformer tap ratio, phase shifter phase shift, and switched shunt nominal Mvar. Various quantities include bus voltage magnitude and angle, bus total generator Mvar injection, generator Mvar injection, branch MW and Mvar flow, and interface MW and Mvar flow.
- [Time Step Simulation](#) Enhancements
 - Added the [storage](#) of the Custom Inputs to the TSB file format.
 - Added the [storage](#) of the Custom Results for Loads to the TSB file format.
 - When using the column plotting option to plot time points, time scale will now be in seconds if no difference between time points is greater than one minute.
 - Modified the Time Step processing when using the SCOPF algorithm at each time step so that less messaging is sent to the Time Step dialog regarding the progress of the SCOPF solution. All the messaging to status edit boxes on the Time Step dialog was greatly slowing down the solution process.
- Transient Stability
 - Transient Model Modification
 - Added ability to specify whether to use the [Runga-Kutta Order 2 \(RK2\) or Euler's method](#) for the integration time step.
 - Added a new option to specify the type of [Exciter Saturation Model](#) function. Choices are Quadratic, Scaled Quadratic, or Exponential.
 - Transient Limit Monitoring
 - Modified the ability of [Action To Take](#) option to allow an option "Trip (Open) Device". The option will trip/open a device" if a monitoring violation occurs (this works for Buses, AC Lines, Generators, Loads, and DC Lines).

- Transient Plots
 - Added the ability to add to a [Plot a Title Block](#) including an image logo, text, and a date.
 - Added the ability to specify special strings such as "@CTGName" which will automatically determine an appropriate string to place in the Plot Title Block Memos, Axis Title captions, Chart Title, or Chart Footer. @CTGNAME will display the appropriate contingency name; @CASENAME will show the name of the case presently open; @BUILDDATE will show the Simulator patch build date; @DATETIME will show the present date and time; @DATE will show the present date; @TIME will show the present time.
 - Added the ability to automatically [save plot images](#) during a transient stability run. The images will be saved in the same directory as the defined hard-drive storage directory.
- [Transient Stability File Format Support](#)
 - Modified the loading of GE DYD files to also accept a file extension of "DYC". When loading a DYC file no prompt will appear asking about deleting existing stability models. The assumption is that it will append to existing models.
 - Added the ability to read a "DYR file with options". When choosing this, you may specify a MCRE *.rwm file to split up generators, a MTRLD *.dat file to split up motors, a GNET *.idv file to disable generator models, and a BASEGEN *.dat to specify the Governor Response Limits flag for generators.
 - Added the ability to load the MCRE, MTRLD, GNET, and BASEGEN files each separately if desired.
 - When writing out a DYR file, we now automatically write out a GNET *.idv and a BASEGEN *.dat file. The filename and path used will be the same as for the DYR file, except "_GNET" and "_BASEGEN" are appended to the filenames.
- Transient Stability Contingency Definitions
 - Added a new [transient contingency event](#) for "OPEN BUS" which opens all AC lines connected to a bus.
 - Added ability to open only one end of a transmission branch in the [transient contingency event](#).
 - Added the ability to assign a model criteria to a [transient contingency action](#). The action will then only be applied if the initial conditions meet the model criteria.
 - Added ability to define [transient contingency definitions based on an injection group](#). The only action allowed is Open and the user must specify whether to open generation, load or shunts. In addition they must specify the amount to open. When the contingency action is implemented the amount to open will be based on the initial condition of the device (not the transient condition), and devices will be opened in order of highest participation factor to lowest participation until at least the amount specified has been exceeded.
- [Transient Stability Script Actions](#)
 - TSSolveAll - Solves all specified transient contingencies.
 - TSSolve("ContingencyName", [StartTime, StopTime, StepSize]) - Solves only the specified transient contingency. StartTime, StopTime, and StepSize are specified in seconds.
 - TSWriteOptions("FileName", [opt1, opt2, ... opt5], KeyField) - Save the transient stability option settings to an auxiliary file specified by FileName. Opt1 through Opt5 should be set to YES or NO according to the specific option categories to save. These parameters are optional and the default is YES for any parameter not specified. opt1 - Save Dynamic Models; opt2 - Save Stability Options; opt3 - Save Stability Events; opt4 - Save Results Settings; opt5 - Save Plot Definitions; KeyField – Optional parameter. Primary, Secondary, or Label. If not specified set to Primary by default.
 - TSCalculateSMIBEigenValues - Calculate single machine infinite bus eigenvalues. Initialization to the start time is always done before calculating eigenvalues.
 - TSSaveTwoBusEquivalent("PWBFileName",[BUS busnum]) - Save the two bus equivalent model of a specified bus to a PWB file. Initialization to the start time is always done before saving the two bus equivalent. "PWBFileName" – name of the PWB file to which to save.

- Bus can be specified by number [BUS busnum], name/nominal kV combination [BUS "busname_nominalKV"], or label [BUS "buslabel"].
- TSLoadGE("FileName",GENCCYN) - Loads transient stability data stored in the GE DYD format. "FileName" – name of the DYD file to load. GENCCYN - YES or NO. Set to YES to split combined cycle units. Set to NO to leave them alone.
 - TSLoadBPA("FileName") - Loads transient stability data stored in the BPA format. "FileName" – name of the BPA file to load
 - TSLoadPTI("FileName", "MCREfilename", "MTRLOfilename", "GNETfilename", "BASEGENfilename") - Loads transient stability data in the PTI format. "FileName" – name of the DYR file to load. This is NOT an optional field. All of the files listed after are optional. If not loading these particular files, specify an empty string, "".
 - Transient Stability Storage
 - [Added support for storing interface](#) MW, Mvar, and MVA flows as part of the transient stability results.
 - Added ability to store switched shunt, to store injection group MW and Mvar outputs, and to store multi-terminal DC record results in transient stability
 - When choosing to [store results to the Hard Drive](#), previously if no directory was specified to which results should be saved then results would not be saved to hard drive at all. Now if the directory is blank, Simulator will default to save data in the same directory as the case was loaded from. Note that if an invalid directory is specified it will offer the opportunity to create the directory or abort the run.

Help

On-line help is available in PowerWorld Simulator via the **Help** ribbon group on the [Window](#) ribbon tab, or by pressing the **F1** key on many dialogs and displays. Context-sensitive help is available on the oneline diagrams. To obtain object specific help, position the mouse over the object in question on the oneline and press the **F1** key.

Sample power flow cases and other information are available at the PowerWorld web site:
<http://www.powerworld.com>

Contact technical support at support@powerworld.com for answers to your questions regarding any PowerWorld product. Or call us at (217) 384-6330.

PowerWorld Corporation

2001 S. First Street

Suite 203

Champaign, IL 61820

Contact Information

PowerWorld Corporation
2001 South First Street
Champaign, IL 61820

Phone: (217) 384-6330
Fax: (217) 384-6329
Toll Free: (877) 748-7840 - U.S. and Canada only

General Information: info@powerworld.com
Technical Support: support@powerworld.com
Sales: sales@powerworld.com

www.powerworld.com

Windows Basics

Operating System

Simulator runs under all Windows XP and later Microsoft Windows Operating Systems.

Mouse Conventions

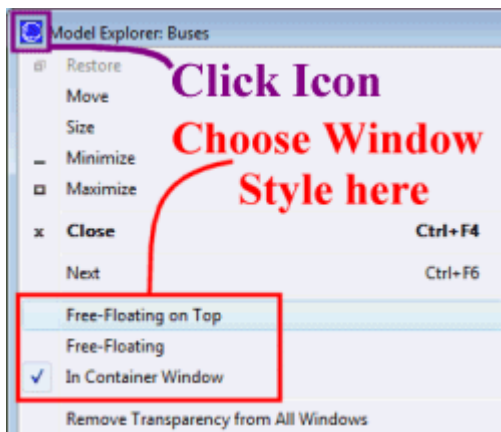
Since much of the interaction between Simulator and the user is accomplished by using the mouse, we have designed the interface to obey consistent conventions for mouse usage. In general, the left-mouse button is used to affect some sort of immediate change or control over a power system element, while the right mouse button is used to gain more information about a power system element or to view a list of available options. More details on are provided throughout this manual.

Dialog Window Position, Size Tracking

As various dialogs are opened, PowerWorld will keep track of the Free-Floating/Contained style as well as the position and size of each dialog. When that dialog is then opened again it will open to the same style, size, and position as it was previously. These settings are stored in the Windows registry so that they are remembered when the program is opened again. You can always reset all these settings to their factory defaults by clicking the **Reset to Default** button on the Window Ribbon Group in the [Windows Ribbon Tab](#).

Dialog Window Styles (Free-Floating, Free-Floating On Top, and In Container)

By default, most dialogs or windows when opened are contained inside the main program window. (The only general exception to this are dialogs that are opened as "modal" windows which you must close before continuing to use the software.) You can change this default globally by choosing **Switch to Free-Floating Windows** on the Window Ribbon Group in the [Windows Ribbon Tab](#). Alternatively, you can leave most windows in the container and individually choose to make specific windows Free-Floating. This is done by left clicking on the icon in the upper left of the dialog to open the windows system menu. This is allowed for all windows except for online diagrams. (Online diagrams always appear inside the main container window.) For windows that we allow this for, three choices will appear as shown below.



In Container Window

In Container Window is the same as the default behavior for most windows. The dialog will be contained inside the main program shell and can not be moved outside of it.

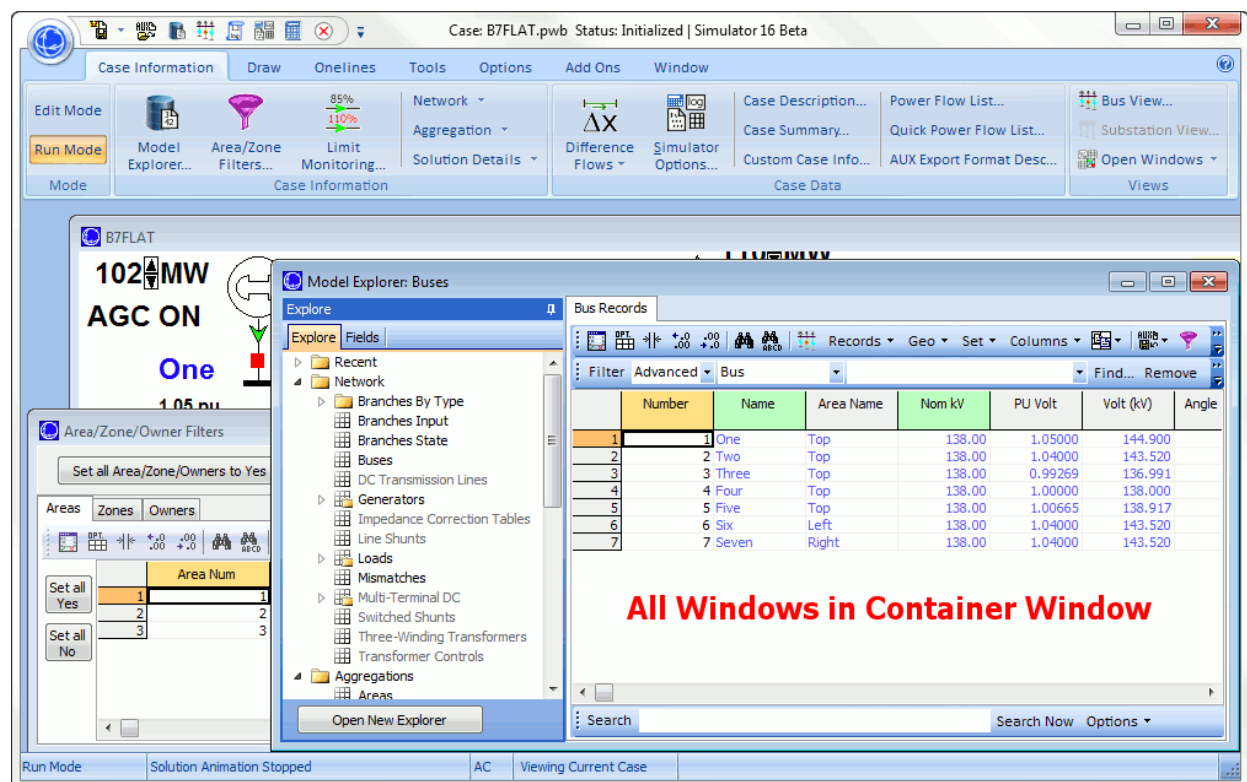
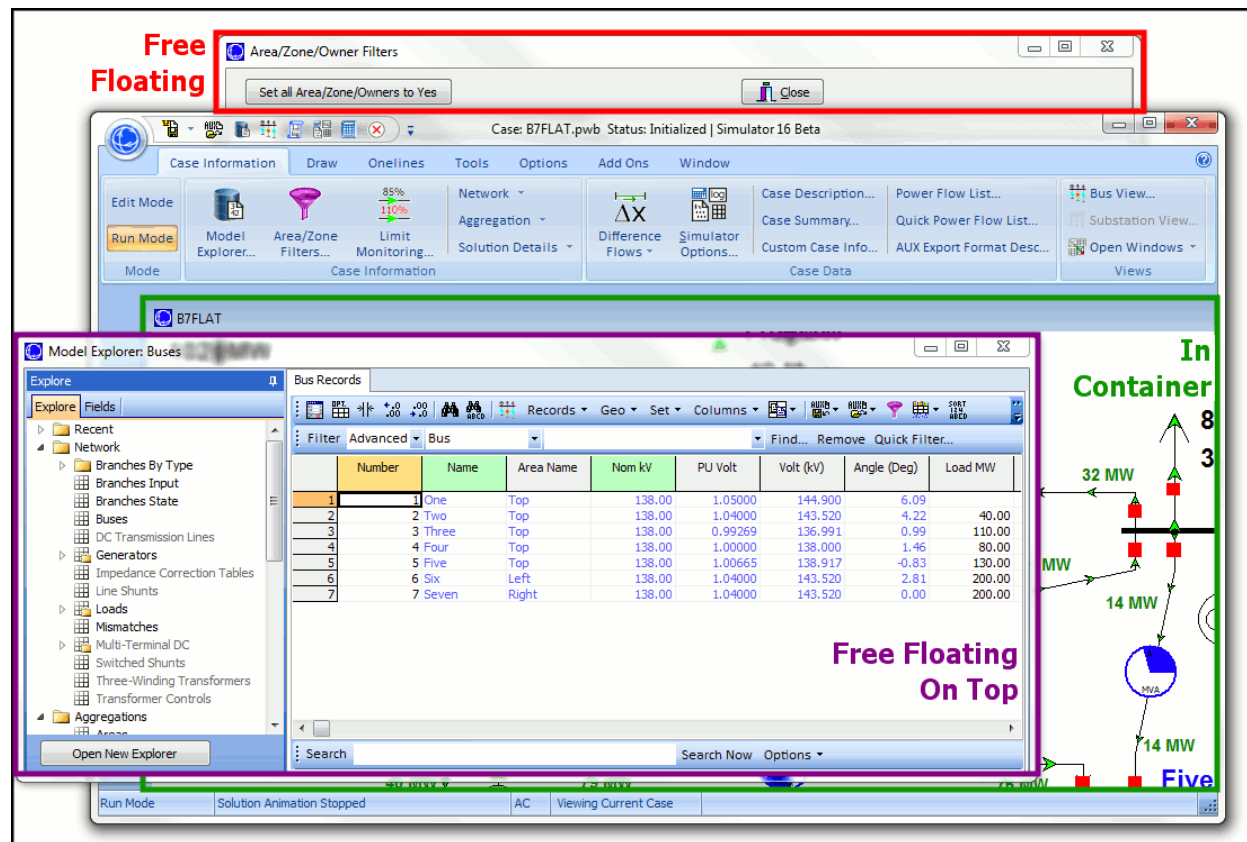
Free-Floating on Top

Setting a window or dialog to Free-Floating on Top will make the window float independent of the main program shell. The dialog will also always remain on top of the main program shell and thus can not get lost behind the main program shell. The [message log](#) window is one of a small number of windows that defaults to this behavior.

Free Floating

This style is the same as Free-Floating on Top, however the dialog will become hidden behind the main program window when the main program window becomes active.

Example Windows Styles



PowerWorld Simulator: Getting Started

The key to using Simulator is to recognize that it has two distinct modes **Edit Mode** and **Run Mode**. The Edit Mode is used to construct new simulation cases or to modify existing cases, while the Run Mode is used to perform the actual power system simulation. You can easily switch between the modes using the **Edit Mode** and **Run Mode** buttons in the **Mode** ribbon group.

If you are new to Simulator and seek a quick means of familiarizing yourself with it, we recommend starting with the tutorials; see [Creating a New Case](#) or [Starting with an Existing Case](#)

Sample cases are provided with the software in the "Sample Cases" directory. If you're interested in learning by doing, you may wish to just [open one of the sample cases](#) and try the different Simulator features.

Edit Mode Introduction

The Edit Mode is used to create a new case or to modify existing cases. To switch to Edit Mode, click on the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

Here is a sampling of things you can do in Edit Mode:

- Create a new case; see [New Case](#) for details.
- Create a new oneline diagram; see [New Oneline](#) for details.
- Add new components graphically to an existing case; see [Draw](#) ribbon tab.
- Modify the appearance of the oneline objects; see [Formatting](#) ribbon group for details.
- View and modify a case using non-graphical lists displays; see [Case Information Displays](#) for details.
- Equivalence a case; see [Equivalencing](#) for details.
- Append a subsystem to an existing case; see [Appending a Case](#) for details.

For more details on the Edit Mode please see [Edit Mode Overview](#).

Run Mode Introduction

The Run Mode is used to solve a single Power Flow Solution, run one of the available load flow tools, or run a time-domain simulation of the power system. To access the Run Mode, click on the **Run Mode** button in the **Mode** group of the Simulator Ribbon.

The key menu associated with the Run Mode is the [Simulation](#) menu. This menu allows you to perform a single Power Flow Solution (however, it is quicker to use the [Quick Access](#) toolbar).

Other key components of the Run Mode include:

- The oneline diagrams, which allow you to view the case graphically. See [Online Diagram Overview](#) for details.
- The Case Information Displays, which allow you to view the entire power system case using list displays. See [Case Information Displays](#) for details.
- Dialogs to change the simulation options and the Power Flow Solution. See [Simulation Options](#) for details.
- Scaling to allow easy variation in the load, shunts, and generation at any number of buses. See [Scaling](#) for details.
- Contouring, which shows a color contour representing the variation in any power system parameter across a system. See [Contouring](#) for details.
- Transfer distribution factor calculations. See [Power Transfer Distribution Factors](#) for details.
- Perform a fault analysis. See [Fault Analysis](#) for details.
- Run [Transfer Capability](#) studies.
- Perform an [Optimal Power Flow](#) (OPF) or [Security Constrained Optimal Power Flow](#) (SCOPF) analysis.
- Generate [PV and QV curves](#).

Script Mode Introduction

The Script Mode is used to access the Script Command window. Simulator scripting allows a method of grouping multiple commands for sequential processing by Simulator. From the [Script Command Execution](#) window, the user can manually enter [script commands](#) for processing, or load an [auxiliary file](#) containing multiple script commands and data modification commands.


Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language. If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to **EDIT** mode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to the **RUN** mode.

To switch modes, use the [EnterMode \(mode\)](#) script command.

There are also a number of general script actions that can be executed in either mode. See [Script General Actions](#) for more details.

Message Log

The Message Log displays detailed results of each Power Flow Solution, chronicling the solution process iteration by iteration. It also reports messages raised by Simulator in performing various operations, such as opening or validating a case. The Message Log can be helpful when you run into problems solving a particular simulation case. The Message Log is not used with Viewer.

To display the Message Log, go to the [Tools](#) ribbon tab, and click the button  **Log** on the **Log** ribbon group. This button is also available on the [Add Ons](#) ribbon tab.

The Message Log is one the few windows that is set by default to be **Free-Floating On Top**. This means that the dialog floats independently of the main program window and always remains on top of the main container window. If you would like to change this style to make the message log contained inside the container window, see the [Window Basics](#) help for more information.

Message Log Local Menu

Right-clicking on the Message Log displays its local menu. To change the font characteristics of the log, select **Change Font**.

To suppress or change color of [certain messages associated with the power flow case](#) select either **Suppress Messages** or **Color Messages** and then the desired message whose characteristics want to change. For more information about the colors and messages see the [Message Log Options](#).

You can find certain text by clicking on **Find**, and then entering the text in the dialog displayed. Also, you can highlight all the text in the log, or inversely unselect all the highlighted text by clicking on **Select All**, or **Unselect All**, respectively. To print or copy the contents to the Windows clipboard of a highlighted section of the message log, select **Print Selection** or **Copy Selection to Clipboard**. You can also clear the contents of the log by selecting **Clear Window**. Lastly, you will find some log options available, such as **Change Maximum Lines**, **Disable Logging**, **Disable Time Stamp**, and **Save Log to File** options which allow automatically saving the log to a file during load flow analysis.

Message Log Color Outlines and Color Background

When solving the power flow, there are several loops which occur that are discussed the [Solving the Power Flow](#) Topic. These loops will generate **Red**, **Purple**, **Green**, and **Blue** outlines on the message log to visually indicate which loop the message text is generated by. This helps in determining which part of a the power flow solution may be having difficulty solving.

Additionally, when the various tools and add-ons send message text to the log the background of the text will be colored to indicate this. For instance, contingency analysis actions will be highlighted with a **light blue** background, while PV, QV, ATC, and OPF add-on message text will be highlighted with a **light yellow** background.

Custom and Memo Display

The Custom tab is now available on many dialogs and displays in Simulator. The purpose of the custom tab is to allow the user to add their own custom information and comments to data and objects in the load flow case. Simply switch to the Memo or Custom page on a display and start typing in the memo box, or add values or text in the custom fields. The information added to these memo pages is stored with the objects in the load flow case and saved to [PWB](#) files.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the object. Defining custom fields is detailed in [Custom Field Descriptions](#).

The custom section also contains a **Selected?** check box. This is a special field that is associated with most objects in Simulator. In many of the places where filtering is allowed, filtering can be done based on the **Selected?** field. Additional information about this field can be found under the [Set Selected Field](#) topic.

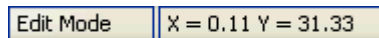
The memo section of the dialog is simply a location to log information about the object. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Status Bar

The Status Bar is displayed across the bottom of the PowerWorld Simulator window. The left-most field of the status bar displays the current Simulator mode (Edit or Run). The remaining fields vary depending on which mode of operation Simulator is currently in. The status bar displays Tool Tips when the cursor is positioned over a toolbar button or menu item. The tool tip is also shown next to the cursor after a short time delay.

Edit Mode

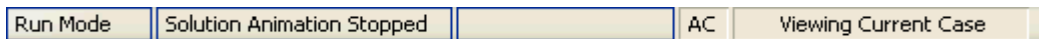
The Edit Mode status bar displays the [Screen Coordinates](#) of the cursor when positioned over a oneline diagram.



Edit Mode Status Bar

Run Mode

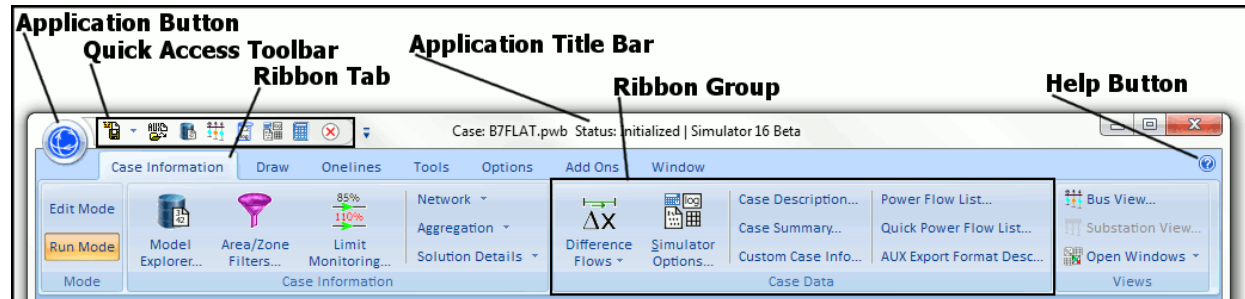
The left side of the Run Mode status bar displays simulation status ("Solution Animation Stopped" or "Solution Animation Running"), AC or DC depending on solution options and the [Difference Flows](#) status ("Current Case", "Base Case", or "Difference Case").



Run Mode Status Bar (Left Side)


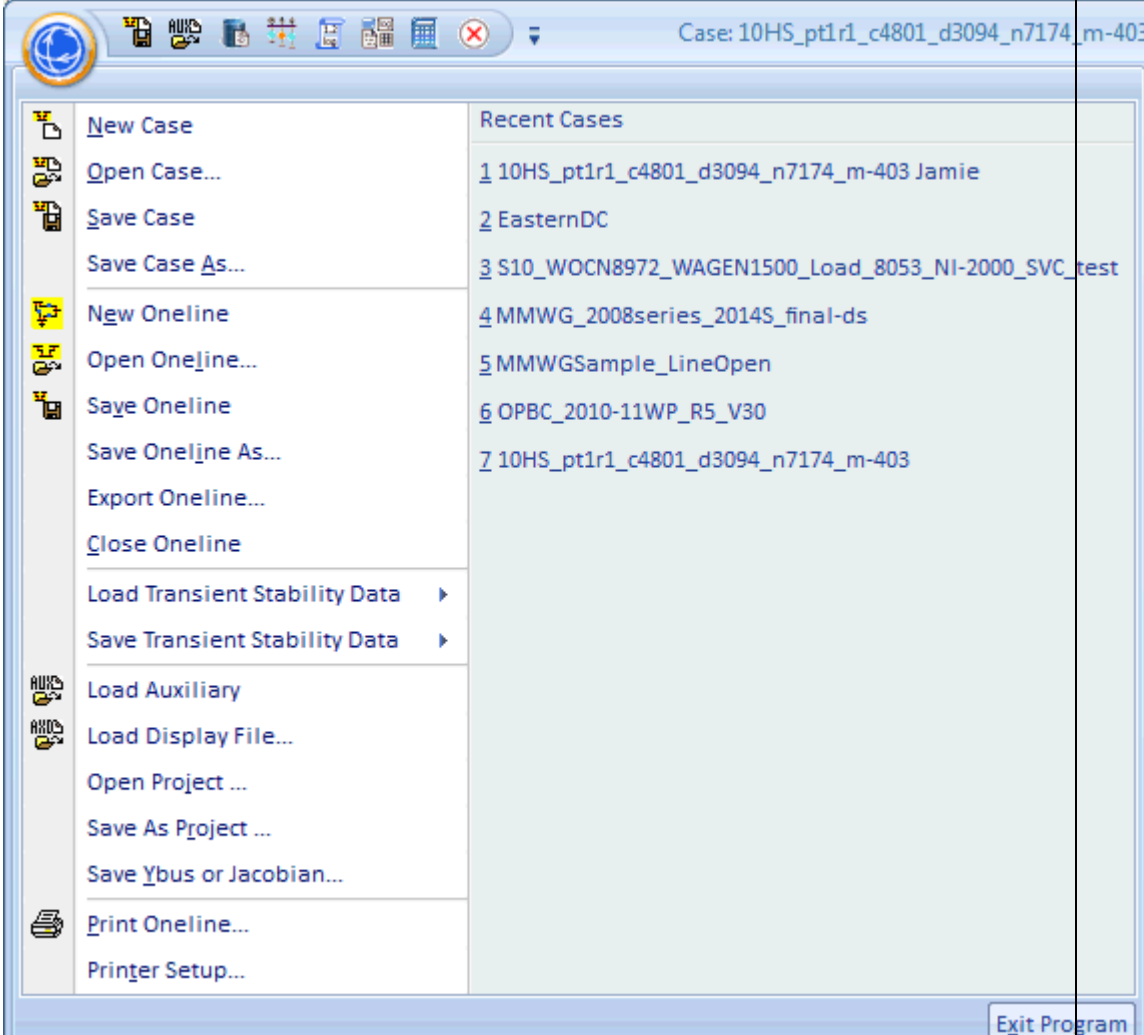
Ribbons User Interface

PowerWorld's user interface consists of a ribbon across the top of the program. Important terminology regarding the ribbon is depicted in the images below.



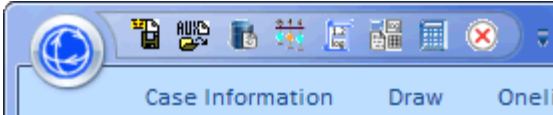
Ribbon	The entire strip across the top is called the ribbon
Ribbon Tab	<p>The ribbon consists of several ribbon tabs which group together features of the software that are used together. There are seven ribbons which have their own purposes and own help topics listed as follows.</p> <ul style="list-style-type: none"> • Case Information Ribbon Tab: primarily used to navigate and look through all the data in your model. • Draw Ribbon Tab: primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in Edit Mode. • Onelines Ribbon Tab: primarily used after you have already created a oneline diagram. This ribbon provides features for customizing the appearance of your oneline diagram. • Options Ribbon Tab: all of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place. • Tools Ribbon Tab: this ribbon provides access to all of the analysis tools that are available in the base package of PowerWorld Simulator. You will use this ribbon when you are performing power flow analysis, contingency analysis, or using the sensitivities tools for instance. • Add Ons Ribbon Tab: this ribbon provides access to all the add-on tools available for Simulator including the OPF, SCOPF, ATC, and PVQV tools. If you have not purchased these tools then the options will be grayed out. • Windows Ribbon Tab: this ribbon provides access to customizing the Windows in the User Interface. It also has some information reading help topics . <p>The goal in designing these ribbons was to create several task specific ribbons which represent the various ways that the software is used. Our hope is that you will not spend a lot of time flipping back and forth between ribbons. If you find yourself doing this please contact PowerWorld Corporation to help us make the software better for the future.</p>
Ribbon Group	Inside a particular ribbon tab, there are around 4 to 7 groupings called ribbon groups. The following image shows three ribbon groups (Individual Insert, Select, and Formatting) with various buttons and menus available.

	<div> <div> </div> <p>ribbon groups join sets of buttons and menus that are related to one another. The large buttons and icons are used to draw attention to the most commonly used features of the software. All features are still available through drop-down menus in the same manner as was available previously. Our design goal in creating and laying out the ribbons was to provide the new user with insight about using the software, while at the same time giving the experienced user quick access to common features.</p> <table> <tr> <td>Large Button</td><td></td></tr> <tr> <td>Large Button Menu</td><td></td></tr> <tr> <td>Small Button With Caption</td><td></td></tr> <tr> <td>Small Button Menu</td><td></td></tr> <tr> <td>Small Button without Caption</td><td></td></tr> <tr> <td>Menu Only Caption</td><td></td></tr> </table> </div>	Large Button		Large Button Menu		Small Button With Caption		Small Button Menu		Small Button without Caption		Menu Only Caption	
Large Button													
Large Button Menu													
Small Button With Caption													
Small Button Menu													
Small Button without Caption													
Menu Only Caption													
Help Button 	Clicking the help button on the right edge of the ribbon brings up this online help												
Quick Access Toolbar	<p>The quick access toolbar at the top left of the ribbon (near the Application Button) contains small buttons with no caption that are always visible. For more information about how to use and customize this see the Quick Access Toolbar help.</p>												
Applicati	Case: B7FLAT.pwb Status: Initialized Simulator 16												

on Title Bar	
Applicati on Button 	<p>Click on this to bring up the Application File Menu shown below. For more information about using the Application File Menu see the Application File Menu help.</p> 

Quick Access Toolbar

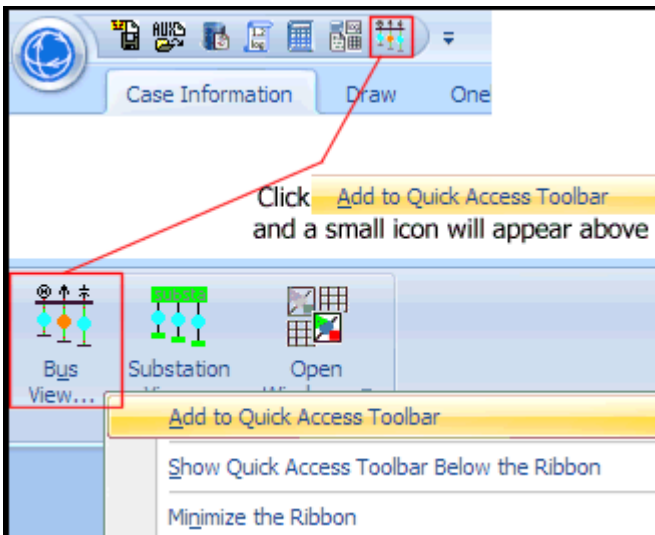
The Ribbon Quick Access Toolbar provides you with quick access to the most frequently used buttons. These buttons are found in the top left portion of the Ribbon just to the right of the [Application Button](#). The default buttons that appear in the Quick Access Toolbar are shown in the figure below.



The eight default buttons are as follows from left to right.

- [Save Case](#)
- [Open Auxiliary File](#)
- [Model Explorer](#)
- [Display Bus View](#)
- [Message Log](#)
- [Simulator Options](#)
- [Single Solution](#)
- [Abort](#)

The Quick Access Toolbar can also be customized very simply. If you find a button that you would like quicker access to, simply right-click on the button and then choose Add to Quick Access Toolbar. The small icon version of the button will then immediately appear up in the Quick Access Toolbar. This is depicted in the image below.

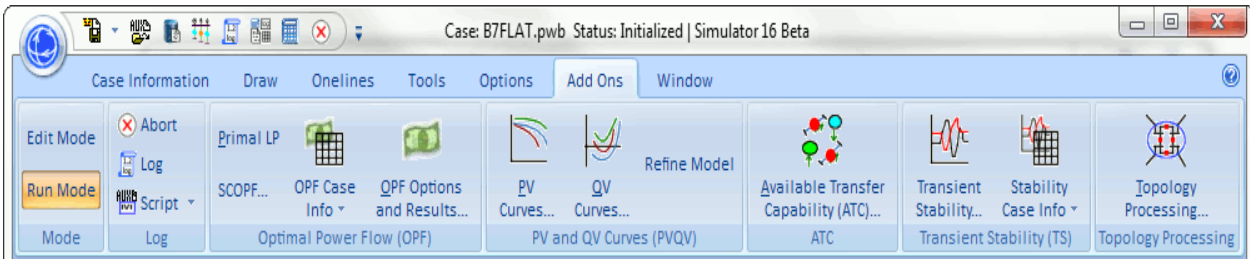


The Quick Access Toolbar can also be modified to show the Toolbar Below the Ribbon providing more space for specifying buttons.

Also experiment with Showing the Quick Access Toolbar below the Ribbon or Minimizing the Ribbon.

Add Ons Ribbon Tab

The Add Ons ribbon consists of buttons and menus that access to the various Simulator Add On Tools. The ribbon is shown below when configured for the OPF, SCOPF, PVQV, ATC, Transient Stability, and Topology Processing add on tools.



The Tools ribbon is broken into seven ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Log Ribbon Group

Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

Optimal Power Flow (OPF) Ribbon Group

Primal LP

Click this button to solve the OPF. A more detailed description of what this does is found in the topic [OPF LP Primal](#).

Security Constrained OPF

Click this button to open the [Security Constrained OPF Dialog](#).

OPF Options and Results

Click this button to open the [OPF Options and Results Dialog](#).

OPF Case Info Menu

The OPF Case Info Menu provides direct access to several case information displays that show default columns related to input and output of the Optimal Power Flow. Instead of using this menu, PowerWorld encourages you leave the [Model Explorer](#) open and navigate using the Explore Pane on the Model Explorer. All of these options are available under the Optimal Power Flow folder on the Model Explorer.



PV and QV Curve (PVQV) Ribbon Group

PV Curves

Click this button to open the [PV Curves dialog](#).

QV Curves

Click this button to open the [QV Curves dialog](#).

Refine Model

Click this button to open the [Refine Model Dialog](#).

Transient Stability (TS) Ribbon Group

Transient Stability

Click this button to open the [Transient Stability Analysis dialog](#).

Stability Case Info Menu

The [Stability Case Info Menu](#) provides direct access to several case information displays that show default columns related to input and output of the transient stability tool.

Topology Processing Ribbon Group

Topology Processing

Click this button to open the [Topology Processing Dialog](#).

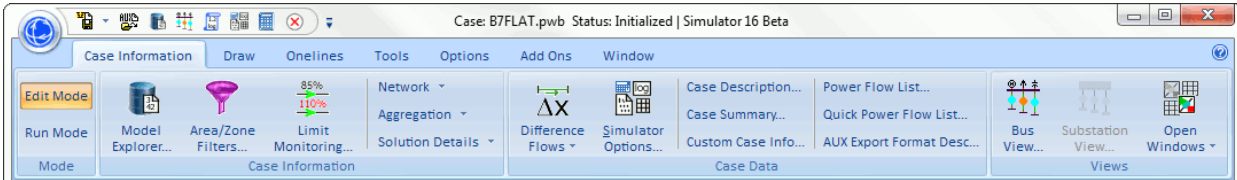
Available Transfer Capability Ribbon Group

Available Transfer Capability

Click this button to open the [Available Transfer Capability Dialog](#).

Case Information Ribbon Tab

The Case Information ribbon consists of buttons and menus that provide access to the information about the power system model. The ribbon is shown below



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Case Information Ribbon Group

Model Explorer



Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

Area/Zone Filters



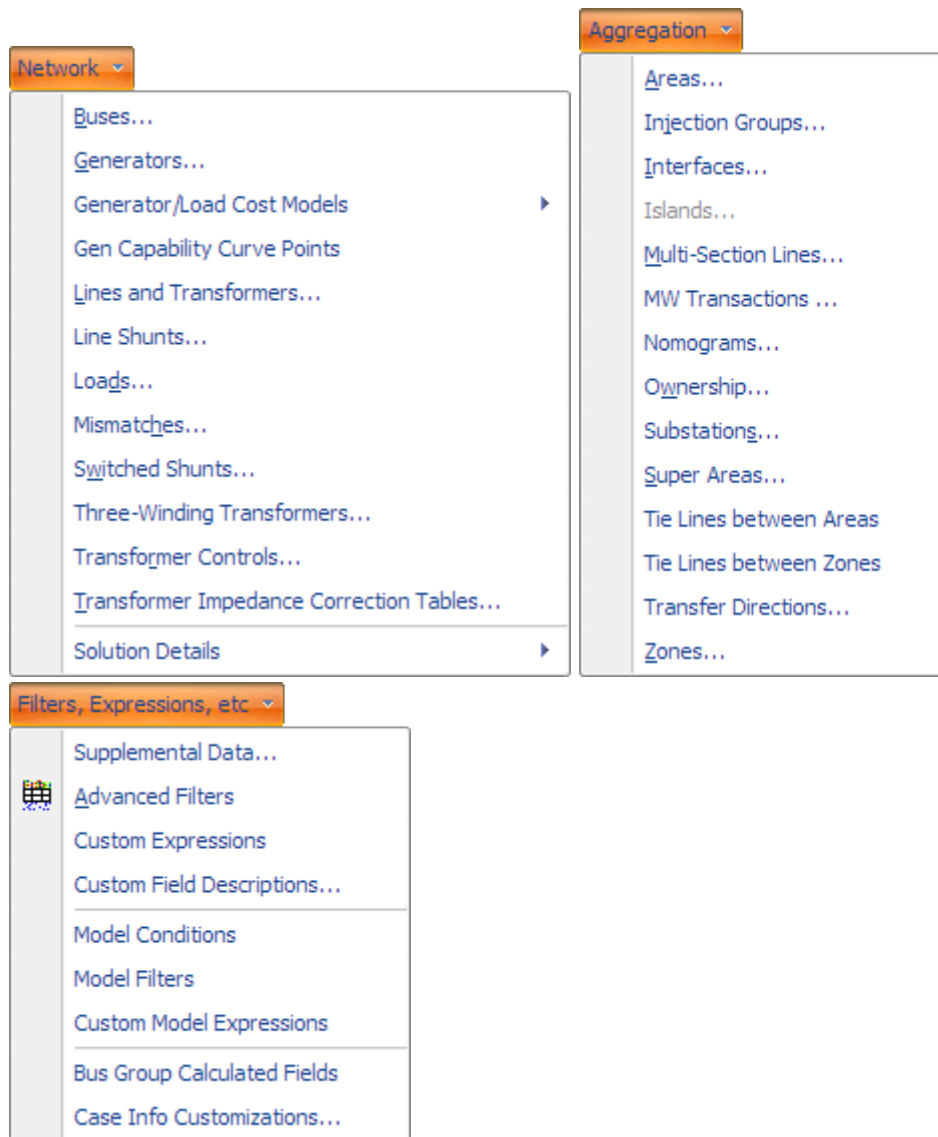
Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.

Limit Monitoring

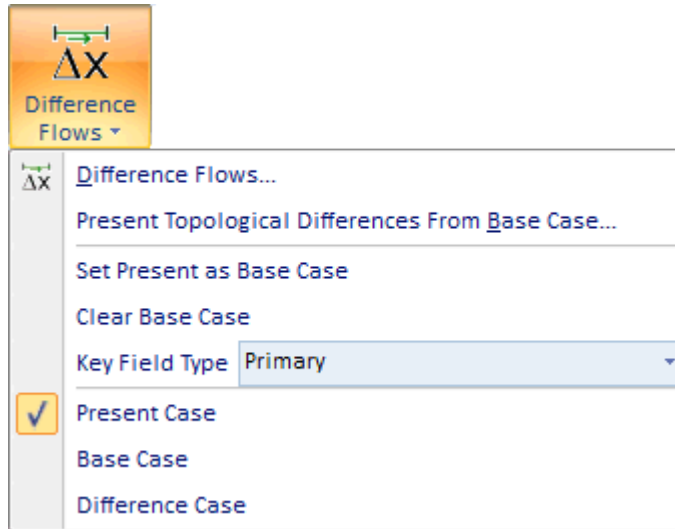
Opens the [Limit Monitoring Settings and Limit Violations Dialog](#) which allows you to specify what to monitor in the various analysis tools.

Menus for Network, Aggregation, and Filters, Expression, Etc

These three menus all perform the same action as opening the Model Explorer, except that they automatically set the Explore Pane of the Model Explorer to show the respective case information display. Instead of using these menus, PowerWorld encourages you leave the Model Explorer open and navigate using the Explore Pane on the Model Explorer.



Case Data Ribbon Group
Difference Flows Menu



Difference Flows: Opens the [Difference Flows Dialog](#)

Present Topological Differences From Base Case: Opens the [Present Topological Differences From Base Case Dialog](#).

The remaining options in this menu all replicate the same behavior available on the [Difference Flows Dialog](#). For more details so that dialog.

Simulator Options

Opens the [Simulator Options Dialog](#).

Case Description

Opens the [Case Description Dialog](#).

Case Summary

Opens the [Case Summary Dialog](#).

Custom Case Info

Opens the [Custom Case Information Display](#)

Power Flow List

Opens the [Power Flow List](#)

Quick Power Flow List

Opens the [Quick Power Flow List](#)

AUX Export Format Desc

Opens the [Auxiliary File Export Format Description Dialog](#) for Power System Objects.

Views Ribbon Group

Bus View

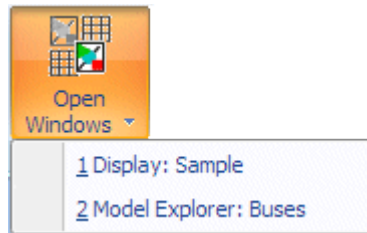
Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Substation View

Opens the [Substation View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

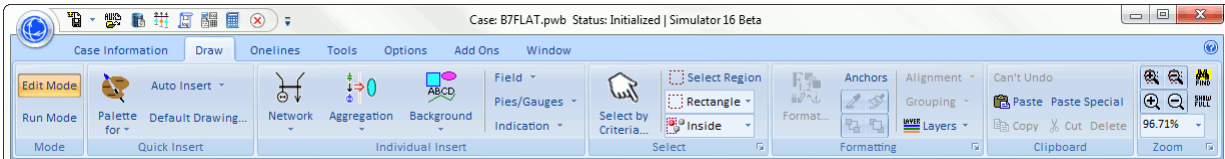
Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Oneline diagrams.



Draw Ribbon Tab

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic.

Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

Clipboard Ribbon Group

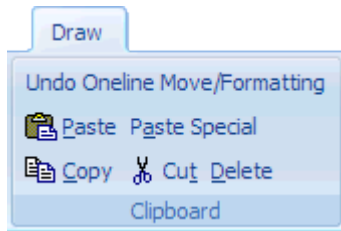
The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.

Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the oneline. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different oneline diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Formatting Ribbon Group

The **Formatting** ribbon group contains options for formatting the selected objects on your active oneline. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.



Format Buttons

Clicking this button will open the [Format Dialog](#). The dialog which opens will have five tabs as follows.

Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the oneline diagram.

Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.

Anchors

Invokes a command to [refresh anchors](#) on all oneline objects.



Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



Send to Back, Bring to Front Buttons

After selecting a group of oneline objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

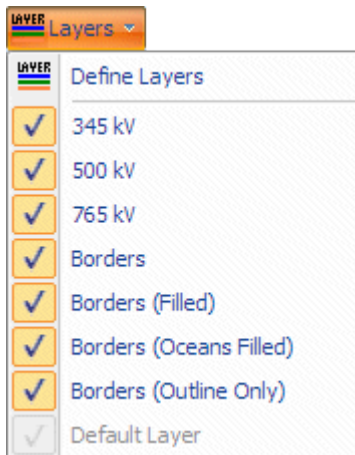
Grouping Menu

Grouping is only available for background oneline objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

Layers Menu

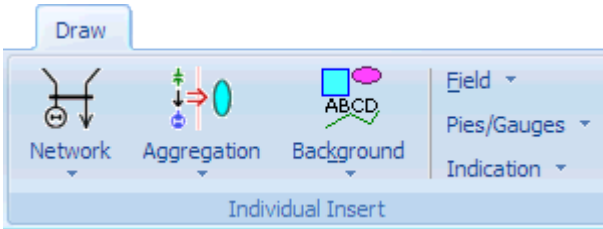
The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert Group](#). The Individual Insert group is shown below.



The six menus available on the group categorize the objects that can be placed on the oneline diagrams. A summary of what is in each menu is described below along with a picture of each of the six menus.

Network Menu

The objects in this menu represent network model objects. Generally these represent actual physical devices such as a generator or transmission line. They are similar to the model objects found in the **Network** Folder on the Explore Pane of the [Model Explorer](#).

Aggregation Menu

The objects in this menu represent model objects that represent an aggregation of other model objects such as interfaces or injection groups. They are similar to the data objects found in the **Aggregation** Folder on the Explore Pane of the [Model Explorer](#).

Background Menu

The objects in this menu represent objects that are only background text and do not link to any of the objects you would look at in the [Model Explorer](#). Some of the objects link to external objects such as documents, but not to elements of the model.

Pies/Gauges Menu

The objects in this menu represent pie charts or gauges which link to the model data found in the [Model Explorer](#).

Field Menu

The objects in this menu represent fields which link to the model data found in the [Model Explorer](#).

Indication Menu

The objects in this menu represent indications of status or flow for objects that are found in the model data found in the [Model Explorer](#).



Network

	Bus
	Generator
	Load
	Switched Shunt
	Transmission Line
	Transformer
	Series Capacitor
	DC Transmission Line



Aggregation

	Area
	Injection Group
	Interface
	MS Transmission Line
	Owner
	Substation
	Super Area
	Zone



Background

	Text
	Memo Text
	Online Link
	Document Link
	Picture
	Background Line
	Background Rectangle
	Background Ellipse
Insert GIS Data From Shapefile	

Pies/Gauges

	Line Flow Pie Chart
	Interface Pie Chart
	MS Line Flow Pie Chart
	Bus Pie Chart/Gauge
	Generator Pie Chart/Gauge
	Switched Shunt Pie Chart/Gauge
	Transformer Pie Chart/Gauge
	Area Pie Chart/Gauge
	Injection Group Pie Chart/Gauge
	Owner Pie Chart/Gauge
	Substation Pie Chart/Gauge
	Super Area Pie Chart/Gauge
	Zone Pie Chart/Gauge
	Old Gauges

Field

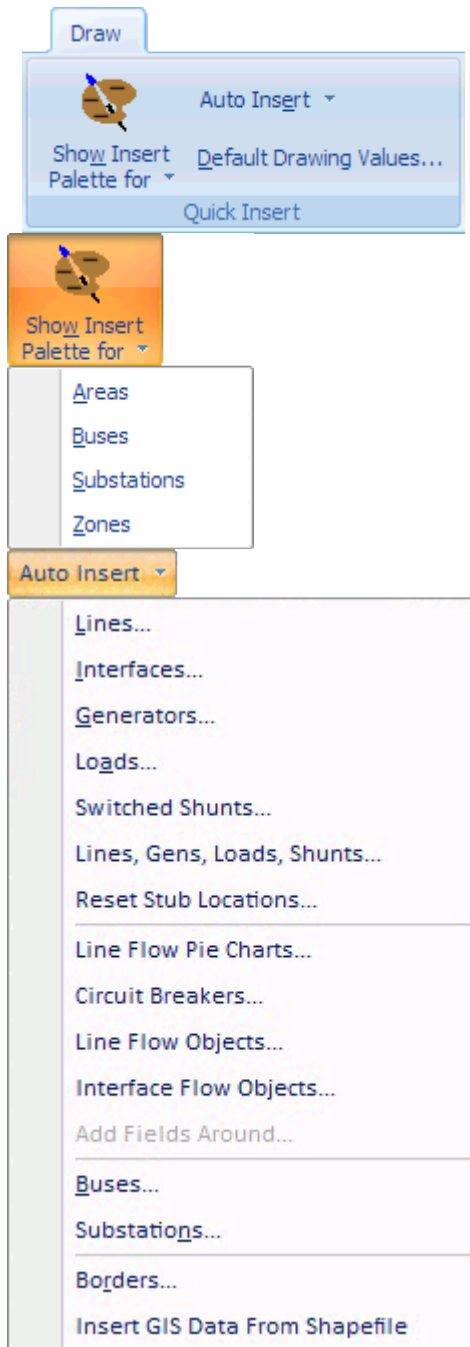
	Bus Field
	Generator Field
	Load Field
	Switched Shunt Field
	Transmission Line Field
	DC Transmission Line Field
	MS Transmission Line Field
	Series Capacitor Field
	Transformer Field
	Area Field
	Interface Field
	Substation Field
	Owner Field
	Zone Field
	Super Area Field
	Case Information Memo Field
	Generic Model Field
	Online Field
	Supplemental Data Field

Indication

	Circuit Breaker
	Line Flow Arrow

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic. Both of the ribbon groups are on the [Draw](#) ribbon tab. The Quick Insert ribbon group is shown as follows.



Show Insert Palette for Menu

Choosing any of the items in the Show Insert Palette for Menu will open up its respective Insert Palette. This Insert palettes show a list of all objects that are already shown on the oneline diagram as well as a list of objects which are not. For more information see the [Using the Insert Palettes](#) help topic.

Show Insert Palette for Menu

The first six options are used to automatically insert network display objects that are connected to bus objects that have already been drawn on the oneline.

Lines : Shows the [Automatically Insert Lines Dialog](#)

Interfaces : Shows the [Automatically Insert Interfaces Dialog](#)

Generators : Shows the [Automatically Insert Generators Dialog](#)

Loads : Shows the [Automatically Insert Loads Dialog](#)

Switched Shunts : Shows the [Automatically Insert Switched Shunts Dialog](#)

Lines, Gens, Loads, Shunts : Shows a dialog for auto-inserting lines, generators, loads, and shunts simultaneously.

Reset Stub Locations : Will completely redo the positioning of objects anchored to buses and substations in the manner done when auto-inserting lines, gens, load, and switched shunts.

The next four options provide methods for adding pie charts, fields, circuit breakers and line flow objects around

objects that have already been drawn on the oneline diagram.

Line Flow Pie Charts : Select this to automatically insert [line flow pie charts](#) on the transmission lines or transformers objects that are selected.

Circuit Breakers : Select this to automatically insert [circuit breakers](#) on the transmission lines or transformers objects that are selected.

Line Flow Objects : Select this option to automatically insert line flow objects on the transmission lines or transformers objects that are selected.

Interface Flow Objects : Select this option to automatically insert Interface flow objects on the interface objects. The Interface Flow Arrow Object which behaves very similar to the Line Flow Object show an arrow to represent the MW flow direction and then shown text for MW value and Mvar value with a sign relative to the MW flow direction.

Add Fields Around : Select this option to insert fields around the presently selected objects. It will then bring up the [Inserting and Placing Multiple Display Objects Dialog](#)

The follow two options provide a mechanism for auto-inserting bus or substation oneline objects. You must have geographic longitude/latitude information in order to use these options.

Buses : Shows the [Automatically Insert Buses Dialog](#)

Substations : Shows the [Automatically Insert Substations Dialog](#)

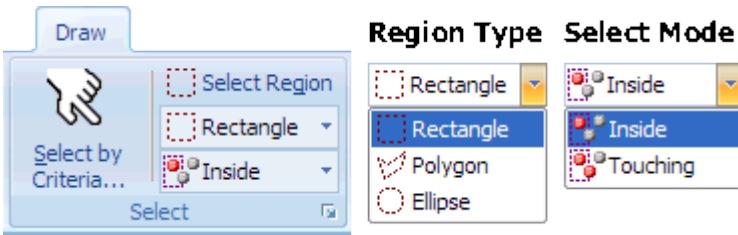
The last two options provide a mechanism for inserting background lines that represent geographic borders or from a Shapefile.

Borders : Shows the [Automatically Insert Borders Dialog](#)

Insert GIS Data From Shapefile : Shows the [Shapefile Import Dialog](#)

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On oneline diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is Polygon, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

Select Mode

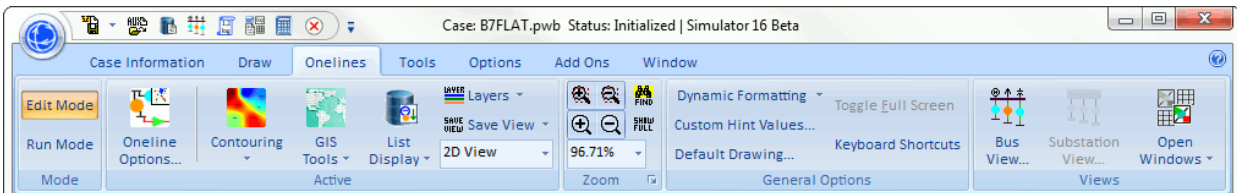
This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key while clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same effect as clicking on the **Select Region** button and selecting a region on your oneline

Onelines Ribbon Tab

The Onelines ribbon tab generally consists of buttons and menus that provide access to modifying the appearance of a oneline diagram you have already created. For access to features regarding adding, moving, resizing, and formatting objects, see the Draw ribbon tab. The Oneline ribbon is shown below.



The Options ribbon consists of buttons and menus that provide access to customizing the software options. The ribbon is shown below.

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Active Ribbon Group

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing oneline diagram. For more detailed help see the [Active](#) ribbon group topic

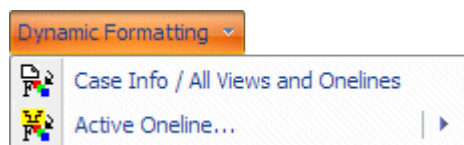
Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

General Options Ribbon Group

Dynamic Formatting Menu

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays.



Custom Hint Values

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

Default Drawing Values

Provides access to the [Default Drawing Values](#).

Toggle Full Screen

Click this button to cause the presently active oneline to be toggled to full screen.

Keyboard Shortcuts

Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

Views Ribbon Group

Bus View

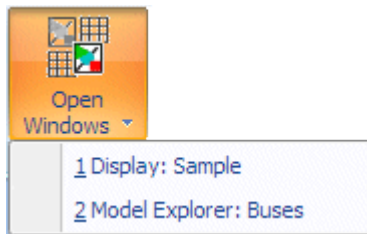
Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Substation View

Opens the [Substation View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

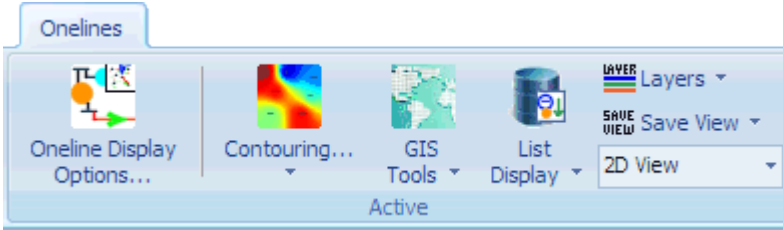
Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Oneline diagrams.



Active Ribbon Group

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing online diagram.



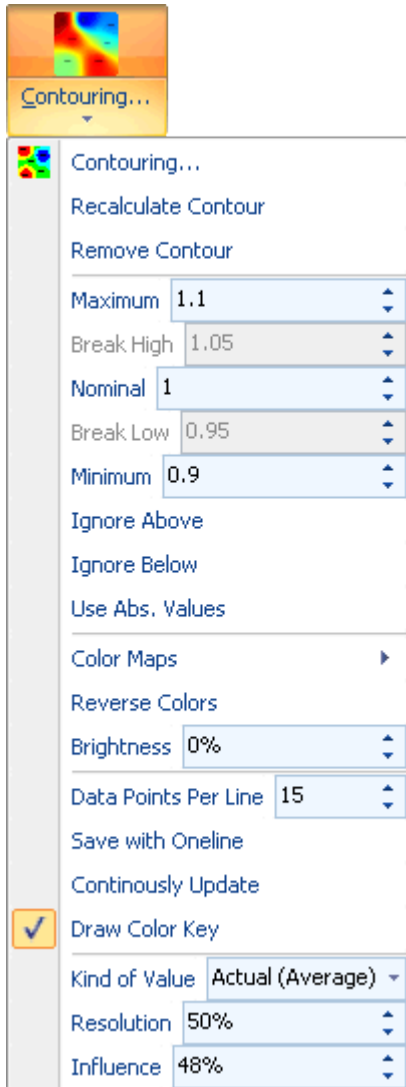
Online Display Options

Opens the [Online Display Options Dialog](#).

Contouring and Menu

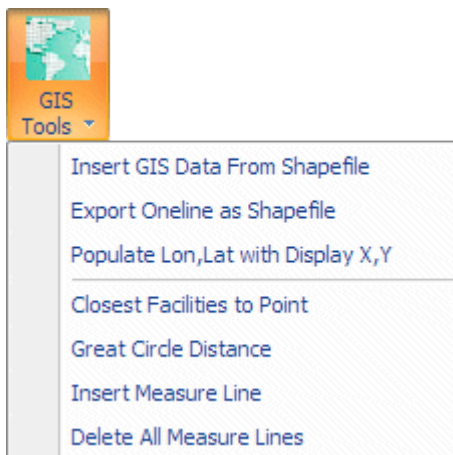
Click the top part of the button to open the [Contouring Dialog](#).

Otherwise, click the drop-down to provide direct access to many of the Contouring Options.



GIS Tools Menu

Provides access to many Geographic Information System related tools that have been built into the software.



Insert GIS Data From Shapefile : Opens the [Shapefile Import dialog](#).

Export Online as Shapefile : Choose this to [Export the Online as a Shapefile](#).

Populate Lon,Lat with Display X,Y : Choose this to populate the Bus records latitude and longitude information based on the locations on the present oneline diagram. For more information see the [Populate help topic](#).

Closest Facilities to Point : Choose this to bring up the [Closest Facility to Point feature](#).

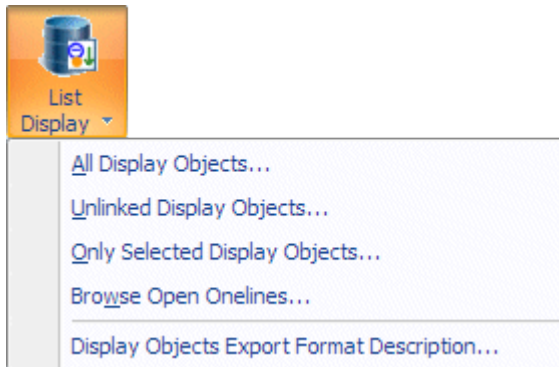
Great Circle Distance : Choose this to calculate the [Great Circle Distance between two points](#).

Insert Measure Line : Provides access to the [Insert Measure Line Feature](#).

Delete All Measure Lines : Provides access to the [Delete All Measure Lines feature](#).

List Displays Menu

Contains access to several ways of viewing the oneline objects inside of "[case information displays](#)". Of course instead of seeing case information, you instead see information about the oneline objects.



All Display Objects: Opens the [Display Explorer](#) providing access to all objects on the active oneline.

Unlinked Display Objects : Opens the [Unlinked Display Objects Dialog](#).

Only Selected Display Objects : Opens the [Display Explorer](#) providing access to objects on the active oneline, but defaults that explorer to only show the presently selected objects.

Browse Open Onelines: Brings up the [Browse Open Onelines dialog](#).

Display Objects Export Format Description : Opens a [Display Objects Export Format Description](#) dialog.

Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Save View

Choose this option to open the [Save Views Dialog](#).

2D View / 3D View

Toggle between 2D and 3D to change the presently active oneline between the 2D and 3D modes.

Zoom Ribbon Group

The Zoom Ribbon Group is on both the [Onelines](#) ribbon tab and the [Draw](#) ribbon tab. To display large detailed power systems, Simulator's onelines possess zooming and panning capabilities. The Zoom Ribbon Group enables you to prescribe a zoom level either by directly specifying a zoom value or by selecting a rectangular region of the diagram on which to focus. There is also a button to open a dialog box from which you can select a bus on which to center the display. See [Zooming and Panning](#) for more information.

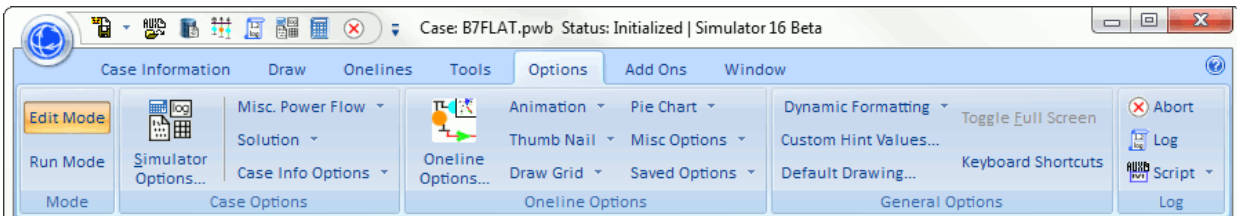


This toolbar contains the following several buttons for zooming the presently active oneline.

- **Zoom Area In /Out** : The top two buttons can be clicked to zoom or out on a particular area of the oneline. When you click these buttons, the cursor will change to a cross-hair waiting for you to choose the region to zoom into (or out of)
- **Zoom In / Out** : The middle two buttons can be clicked to change your mouse into a zooming tool. After choosing the button with a plus sign, each time you click on the oneline you will zoom in on that location. When choosing the button with a negative sign you will zoom out instead.
- **Present Zoom Level**: the number represents the present zoom level and can be manually changed to modify the zoom level.
- **Show Full** : Automatically zooms and pans the presently active oneline diagram so that all objects on the oneline diagram can be seen.
- **Find**: Provide access to the [Find Object on Oneline Dialog](#)

Options Ribbon Tab

The Options ribbon consists of buttons and menus that provide access to customizing the software options. All of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place. The ribbon is shown below.



The Options ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Case Options Ribbon Group

Simulator Options

Opens the [Simulator Options Dialog](#).

Misc. Power Flow Menu

Provides more direct access to some miscellaneous options that are also found on the Opens the [Simulator Options Dialog](#).

Solution Menu

Provides more direct access to some options related to the power flow solution algorithm. These options are all also available on the [Simulator Options Dialog](#).

Case Info Options Menu

Provides more direct access to some options related to the appearance of case information displays. These options are all also available on the [Case Information Display Options](#) on the [Simulator Options Dialog](#).

Online Options Ribbon Group

Online Display Options

Opens the [Online Display Options Dialog](#).

Pie Chart Menu

Provides more direct access to options related to Pie Charts. These options are all also available on the [Online Display Options Dialog](#) on the [Pie Charts/Gauges](#) page.

Animation Menu

Provides more direct access to options related to Animated Arrows. These options are all also available on the [Online Display Options Dialog](#) on the [Animated Flows](#) page.

Thumb Nail Menu

Provides more direct access to options related to Thumb Nail View. These options are all also available on the [Online Display Options Dialog](#) on the [ThumbNail View](#) page.

Draw Grid Menu

Provides more direct access to options related to the Draw Grid. These options are all also available on the [Online Display Options Dialog](#).

Misc. Options Menu

Provides more direct access to miscellaneous options related to oneline diagrams. These options are all also available on the [Online Display Options Dialog](#).

Saved Options Menu

This menu provides direct access to the for toggling between different sets of options. These different sets of options are defined on the [Online Display Options Dialog](#) by clicking on the **Save Options to Case** button on that dialog..

General Options Ribbon Group

Dynamic Formatting Menu

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays._

Custom Hint Values

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

Default Drawing Values

Provides access to the [Default Drawing Values](#).

Toggle Full Screen

Click this button to cause the presently active oneline to be toggled to full screen.

Keyboard Shortcuts

Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

Solution Options Quick Menu

The Solution Options Quick Menu provides access to most of the solution options contained in the [Solution Options](#) dialog in an individual manner through menu commands.

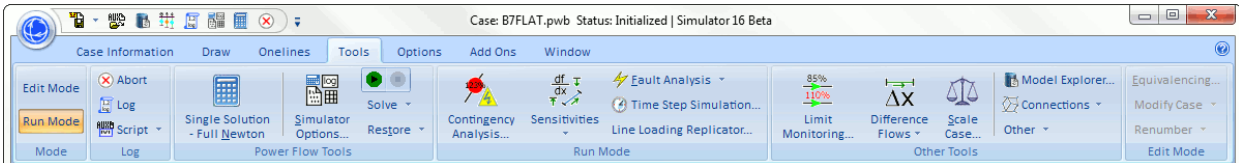


This quick menu contains the following default options:

- Solution Convergence Tolerance (MVA)
- Maximum Solution Iterations
- Maximum Outer Control Loop Iterations
- Enable/Disable Only One Iteration
- Enable/Disable Optimal Multiplier
- Enable/Disable Enforcing Generator MW Limits
- Enable/Disable Automatic Generation Control (ACE)
- Enable/Disable Generator MVAR Checking
- Enable/Disable Checking Generator MVAR Immediately
- Enable/Disable Switched Shunt Control
- Enable/Disable Transformer LTC Control
- Enable/Disable Phase Shifter Control
- Enable/Disable Modeling Phase Shifters as Discrete Controls
- Enable/Disable Preventing Controller Oscillations
- Set Island-Based AGC Control Type
- Set Island-Based AGC Tolerance (MW)

Tools Ribbon Tab

The Tools ribbon consists of buttons and menus that provide access to the analysis tools that come with all versions of PowerWorld Simulator. Note: Features related to the various add-on tools including OPF, SCOPF, PVQV Curve, and ATC are found on the [Add Ons](#) ribbon tab. The tools ribbon is shown below



The Tools ribbon is broken into six ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Log Ribbon Group

Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

Power Flow Tools Ribbon Group

This group represents the basic solution actions for solving the power flow.

For more detailed help see the [Power Flow Tools](#) ribbon group topic.

Run Mode Ribbon Group

The Run Mode Ribbon Group contains basic analysis tools that are only available in [Run Mode](#).

For more detailed help see the [Run Mode Ribbon Group](#) topic.

Other Tools Ribbon Group

The Other Tools Ribbon Group contains basic analysis tools that are normally available in both [Run Mode](#), and [Edit Mode](#).

For more detailed help see the [Other Tools Ribbon Group](#) topic.

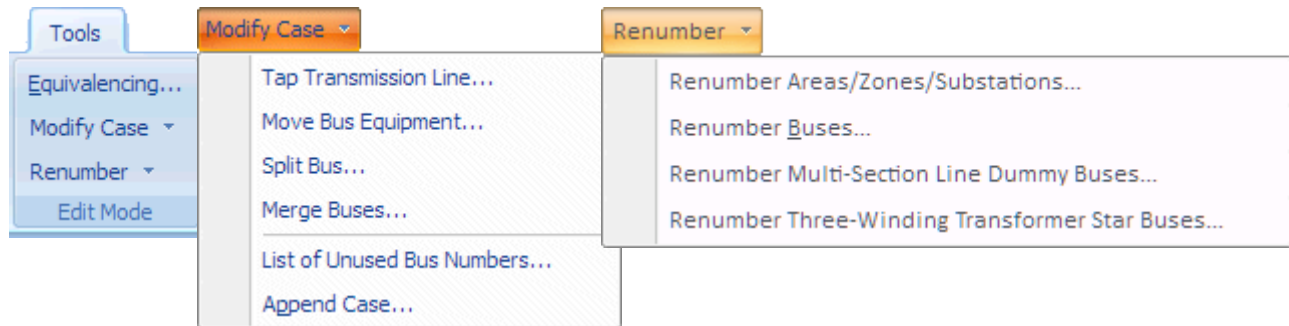
Edit Mode Ribbon Group

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#).

For more detailed help see the [Edit Mode Ribbon Group](#) topic.

Edit Mode Ribbon Group

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#). The Group is shown below.



Equivalencing

Click this button to open the [Equivalencing Dialog](#).

Modify Case Menu

The Modify Case Menu contains actions related to modifying the network model.

Tap Transmission Line: Opens the [Tap Transmission Line Dialog](#).

Move Bus Equipment: Opens the [Move Bus Equipment Dialog](#).

Split Bus: Open the [Split Bus Dialog](#).

Merge Buses: Opens the [Merge Buses Dialog](#).

List of Unused Bus Numbers: Clicking this button to get a [list of unused bus numbers](#) in the model.

Append Case: Provides access to [opening a case while appending](#).

Renumber Menu

The Renumber Menu contains actions related to renumbering data in the model.

Renumber Areas/Zones/Substations: Opens the [Renumber Areas, Zones and Substations Dialog](#).

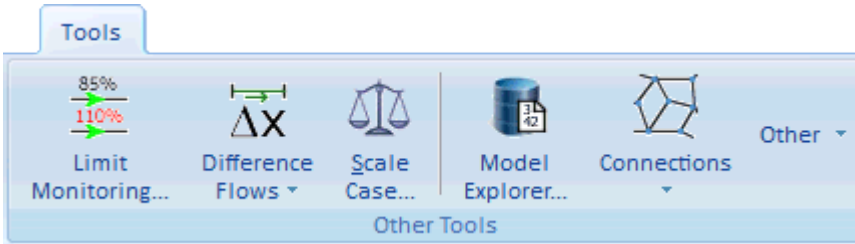
Renumber Buses: Opens the [Renumber Buses Dialog](#).

Renumber Multi-Section Line Dummy Buses: Allows user-specification of dummy bus numbers for multi-section lines. See the section **Multi-Section Line records > Renumber Dummy Buses** with the [Multi-Section Lines Display](#) topic for more information.

Renumber Three-Winding Transformer Star Buses: Allows user-specification of three-winding transformer star bus numbers. See the section **3W Transformers records > Renumber Star Buses** with the [Three Winding Transformer Display](#) topic for more information.

Other Tools Ribbon Group

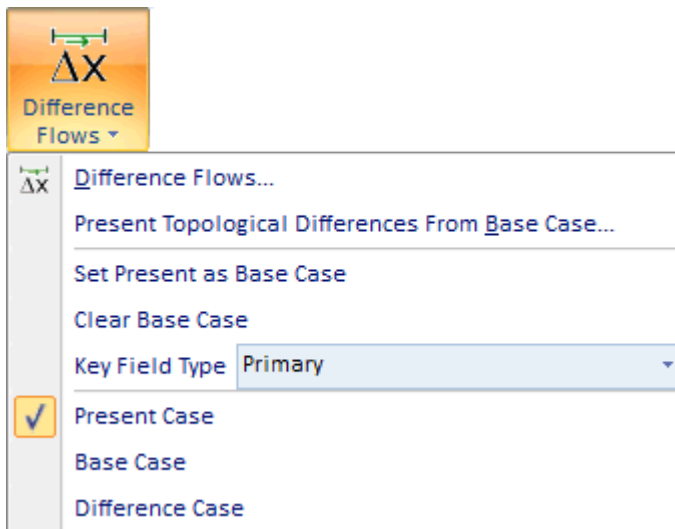
The Other Tools ribbon group contains basic analysis tools that are normally available in both [Run Mode](#) and [Edit Mode](#). The group is shown below.



Limit Monitoring Settings

Opens the [Limit Monitoring Settings Dialog](#).

Difference Flows Menu



Difference Flows: Opens the [Difference Flows Dialog](#)

Present Topological Differences From Base Case: Opens the [Present Topological Differences From Base Case Dialog](#).

The remaining options in this menu all replicate the same behavior available on the [Difference Flows Dialog](#). For more details so that dialog.

Scale Case

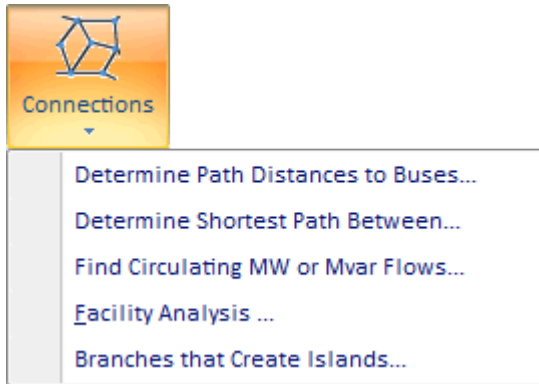
Opens the [System Scaling Dialog](#).

Model Explorer

Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

Connections Menu

The Connection Menus contains actions related to analyzing the interconnectedness of the network model. These may be considered Graph Theory routines.



Determine Path Distances to Buses: Opens the [Determine Patch Distances from Buses or Group Dialog](#).

Determine Shortest Path Between: Opens the [Determine Shortest Path Between Dialog](#).

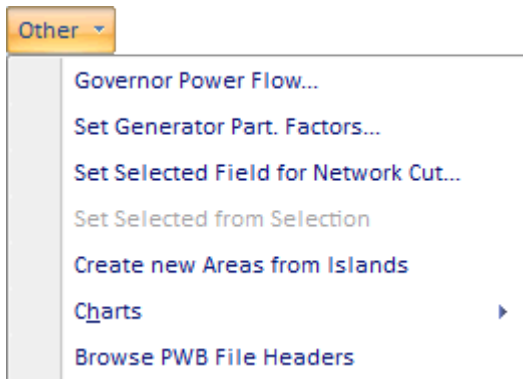
Find Circulating MW or Mvar Flows: Opens the [Mvar and MW Flow Cycle Dialog](#).

Facility Analysis: Open the [Facility Analysis Dialog](#).

Branches that Create Islands: Opens the [Branches that Create Islands Dialog](#).

Other Menu

The Other Menu contains other miscellaneous features.



Governor Power Flow: Opens the [Governor Power Flow Dialog](#).

Set Generator Part. Factors: Opens the [Set Generator Participation Factors Dialog](#)

Set Selected Field for Network Cut: Opens the [Set Selected Field for Network Cut Dialog](#)

Set Selected From Selection: When a selection is chosen in [Edit Mode](#), click this to set the Selected? field to YES for that selection.

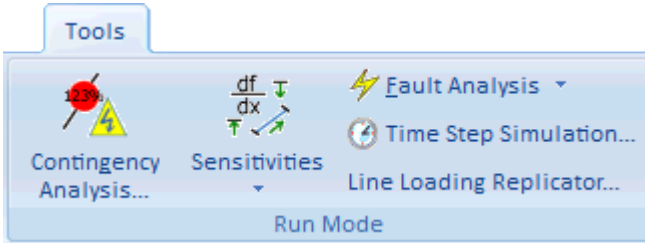
Create new Areas from Islands: When selected, Simulator will replace existing Area designations of the buses in the case with new areas defined by the electrical islands existing in the case.

Charts Menu: The Charts menu is only available in [Run Mode](#). It provides access to various charts including the [Area ACE Chart](#), [Area Load and Generation Chart](#), [Area Losses Chart](#), [Area MW Transactions Chart](#), and [Area Average Cost Chart](#).

Browse PWB Header Files: When selected, Simulator will prompt you to specify a directory containing PWB files for which you wish to preview the header (case description) text. Once the directory has been specified, Simulator will obtain the case description for each PWB file in the directory, and write them all into the Simulator message log.

Run Mode Ribbon Group

The Run Mode ribbon group contains basic analysis tools that are only available in [Run Mode](#). The group is shown below.

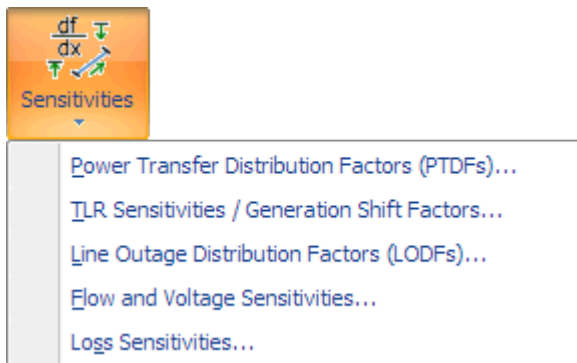


Contingency Analysis

Click this button open the [Contingency Analysis Dialog](#) providing access to the automated processing of a list of contingencies.

Sensitivities Menu

The Sensitivities Menu contains a list that provides access to all the sensitivity calculations available.



[Power Transfer Distribution Factors \(PTDFs\)](#) opens its associated dialog.

[TLR Sensitivities / Generation Shift Factors](#) opens its associated dialog.

[Flow and Voltage Sensitivities](#) opens its associated dialog.

[Loss Sensitivities](#) opens its associated dialog.

Fault Analysis

Click this button to open the [Fault Analysis Dialog](#).

Time Step Simulation

Click this button to open the [Time Step Simulation Dialog](#).

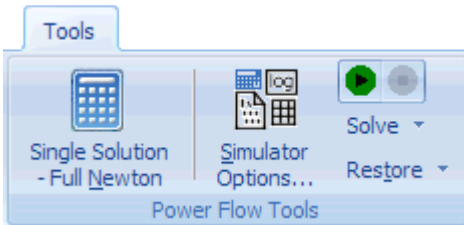
Line Loading Replicator

Click this button to open the [Line Loading Replicator Dialog](#).

Power Flow Tools Ribbon Group

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Simulator Options

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using the either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

Play Button

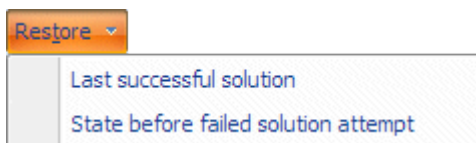
Click this button to cause repeated solutions of the system and to cause the animation of oneline diagrams

Pause/Stop Button

Click this button to stop the action of the Play button.

Restore Menu

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

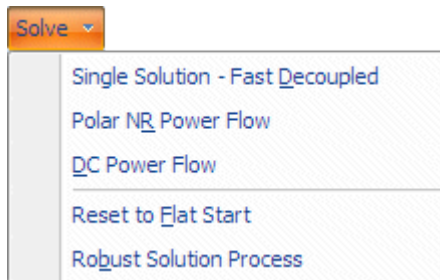
To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

Solve Menu



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Polar NR Power Flow

This performs a Newton Raphson power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

DC Power Flow

This automatically changes the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

Reset to Flat Start

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero.

Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

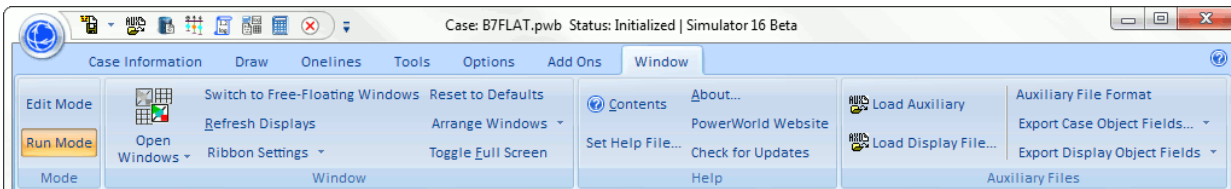
Robust Solution Process

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.

Window Ribbon Tab

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

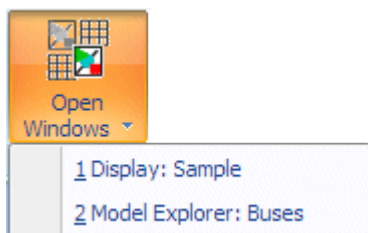
Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Window Ribbon Group

Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



Switch to Free-Floating Windows (Switch to Standard Container Window)

Choosing this option toggles the program from between two different window environment

1. All diagrams and dialogs are contained within the Simulator program shell
2. All diagrams and dialogs are free-floating on your monitor.

For more information see the [Window Basics](#).

Reset to Defaults

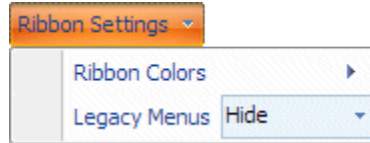
The program keeps track of dialog Free-Floating/Contained Style, Size, and Position. Clicking this button will reset all these to the factory default settings. Nearly all dialogs in will then be reset back to Contained with a default size and position as well. For more information on Style, Size and Position, see [Window Basics](#).

Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

Ribbon Settings Menu

This menu contains options for customizing your ribbon.



Ribbon Colors : You can change the color scheme to either **Blue**, **Silver** or **Black**

Legacy Menus : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

Arrange Windows

The options under Arrange Windows are Tile Horizontally, Tile Vertically, and Cascade. Tile rearranges the open online diagrams such that the total window area is divided equally among all of them and each is completely visible. Cascade Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

Toggle Full Screen

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

Help Ribbon Group

Contents

Opens the [Simulator Options Dialog](#).

Set Help File

Opens the [Case Description Dialog](#).

About

Click this to open the About dialog. The About dialog contains information about the software as well as showing you the Build Date of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About dialog.

There is also an option on the About dialog to Change License Key. This allows selection of a new license key file. Add ons are controlled through the license key. If you are using different versions of Simulator with different add ons, this option provides the mechanism for switching between the different versions of Simulator. You will need a different license key file if you are using different licenses for Simulator that have different add ons.

PowerWorld Website

Click this to open your default web browser to the PowerWorld Corporation Website.

Check for Updates

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

Auxiliary File Ribbon Group

Load Auxiliary

Choose this option to load an [Auxiliary File](#).

Load Display Auxiliary File

Choose this option to load a [Display Auxiliary File](#) to append to the active oneline.

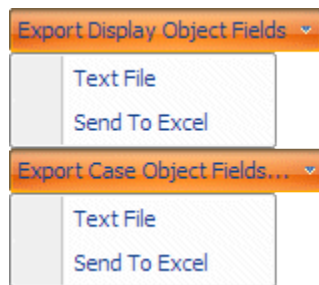
Auxiliary File Format

Opens the [Bus View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Export Display Object Fields Menu

Export Case Object Fields Menu

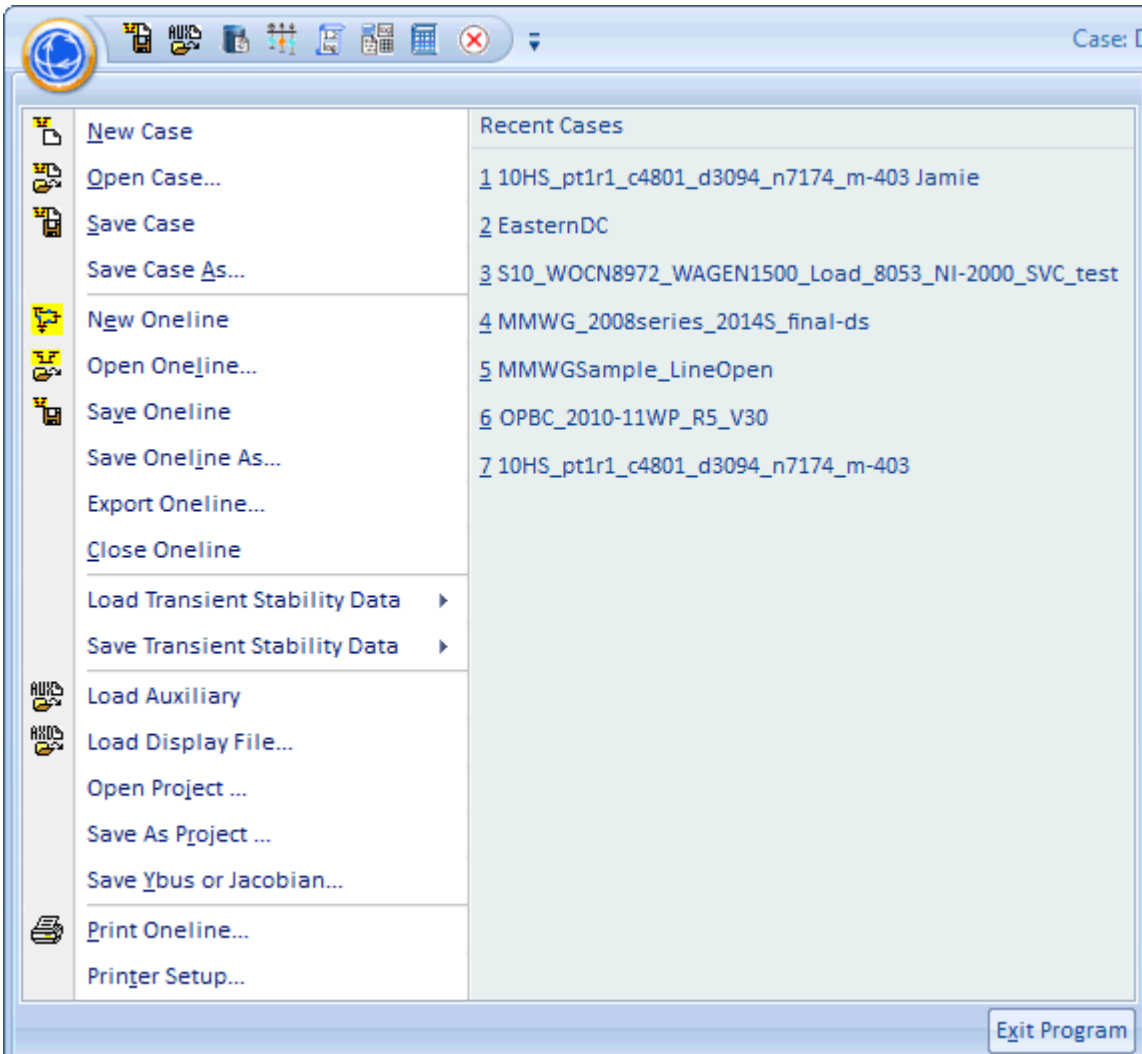
Choose these options to export a list of fields for each type of object in a case (or a Oneline Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



Application File Menu

The Application File menu is used to open and save either full cases or oneline diagrams, to create new cases, to open and load auxiliary files, to print and to exit Simulator. To access this menu you must click on the Application button in the upper-left corner of the program.

Application Button: 



Case Formats

Simulator supports a number of case formats. You can both open and save cases using most of the following formats:

PowerWorld Binary (*.pwb) (Preferred Format)

For most users, the best choice of power flow case formats is the PowerWorld Binary format. This format stores the most complete set of case information but requires the smallest file sizes. Information stored in this format includes power flow data, economic parameters, case time variation/options values, and screen customizations. The only potential disadvantage of this format is that it is stored in binary form, which means that it cannot be viewed using a standard text editor. All the other formats are ASCII and thus readable in text editors such as Notepad.

One additional benefit of using the Powerworld Binary format is that with this format the case can be password-protected. This setting is available under **Misc. Power Flow > Set Case Username and Password** in the **Case Options** ribbon group on the [Options](#) ribbon tab.

PowerWorld Auxiliary File (*.aux)

Simulator can now save and load cases using the PowerWorld defined auxiliary file format. When the aux file format is chosen as the save file type in Simulator, the data describing the entire case is written to a text file described by Simulator's auxiliary file format. These files can be considerably larger than PowerWorld Binary files, and generally should not be used unless otherwise necessary. Note that not all auxiliary files represent an entire case, and in fact in most cases do not. Only when an auxiliary file is created using the **Save Case As** option in the [Application Button](#) in Simulator does the auxiliary file contain all necessary information for loading the complete case in Simulator as an auxiliary file.

PowerWorld Auxiliary File - Secondary Keys (*.aux)

This option is the same as choosing PowerWorld Auxiliary File *except* for the key identifiers for each record. Instead of using the primary keys (i.e. bus numbers) to identify devices in the data, the secondary keys (i.e. combination of bus name and nominal voltage) will be used as the identifiers in the saved file.

PowerWorld Auxiliary File - Labels (*.aux)

This option is the same as choosing PowerWorld Auxiliary File *except* for the key identifiers for each record. Instead of using the primary keys (i.e. bus numbers) to identify devices in the data, the device primary [labels](#) will be used as the identifiers in the saved file.

PTI Raw Data Format (*.raw)

This format is included primarily for interchange of power flow data with other packages. The PTI Raw Data format only contains power flow data. When using this format, the generator participation factors are automatically set to be proportional to the MVA rating of each generator.

When reading a RAW file that has comments on a given line, the comments will be loaded into the Memo field of the corresponding data record. However, the Memo field will not be written back out when saving a RAW file.

PTI Raw Data Format (*.raw) (with options)

This format is the same as the PTI Raw Format except that the [PTI RAW File Load Options](#) dialog will be opened to allow user input for various options.

GE PSLF Format (*.epc)

This format is included primarily for importing load flow data saved from GE's PSLF program. The EPC format is a text file format. This format does contain excess information that Simulator does not

use, however Simulator does store this information for editing and writing back out to EPC format files.

GE PSLF Format (*.epc) (with options)

This format is the same as the GE PSLF Format except that the [GE EPC File Load Options](#) dialog will be opened to allow user input for various options.

Areva HDBExport Format (*.csv)

Customers who use the Areva EMS system are able to use a command in the Areva EMS called HDBExport to save a comma-delimited file containing a complete description of the network model in the EMS system. The CSV file must be exported using appropriate flags and a specific *pattern file* in order to generate the format required by PowerWorld. The resulting file will be a full-topology representation of the network model appropriate for use with the [Integrated Topology Processing](#) add-on. For more information, please contact PowerWorld Corporation offices.

BPA IPF Format (*.net)

The BPA IPF Format is another format which can be read from our customers in China.

IEEE Common Format (*.cf)

The IEEE Common Format is used to specify only power flow information. CAUTION: IEEE common format does not support many of the formats used in the PowerWorld packages, such as multiple loads and generators at a bus. Usually, IEEE Common Format is used only for inputting cases.

PowerWorld Simulator Project Files (*.pwp)

Project files are described in [Overview of PowerWorld Simulator Project Files](#).

Advanced settings for opening cases are available under the [File Management settings](#).

Special Data Handling

Some special handling of data may occur when loading, appending, or saving data in the various formats. The following details what occurs in these situations.

PowerWorld Binary (*.pwb)

Appending

- Appended data completely overwrites an existing record corresponding to the same bus number(s).
- Branch elements are appended to a case only if both of their terminal buses exist.

PTI RAW Data Format (*.raw)

Appending

- Appended data completely overwrites an existing record corresponding to the same bus number(s).
- Branch elements are appended to a case only if both of their terminal buses exist.
- Any new buses or branches that are created are now flagged as having just been added. This flag is then used as part of the pre-processing feature in the power flow solution to automatically estimate appropriate voltage magnitudes and angles for any existing or new branches and buses that become newly energized.

- A field, **EPC File\EPC Modification Status**, is populated to indicate whether an object is new or modified while reading. Even though this is a field that is normally associated with loading in an EPC file, the meaning is the same so the same field is used.

Saving

- When saving a case as a PTI RAW files the switched shunt records will always write out at least one block. If none exists, a single block with 1 step and size equal to the present MVar output is written.

GE PSLF Format (*.epc)

Reading

- If the latitude and longitude values are both zero when loading an EPC file, both entries will be ignored as invalid..
- A designation of the area and zone of a GE branch and transformer will be stored when loading from an EPC file.
- When loading multi-terminal dc lines from an EPC file, current limits for dc converters will not be enforced. This is set via a field, **Enforce Current Limit**, available with dc converters.
- When reading in a GE EPC file, the "aloss" parameter is usually rounded to one bus or the other to be expressed in terms of the "From End Metered" field (except in the case of Multi-Terminal DC converters).
- Simulator does not consider load IDs to be case sensitive. If multiple loads are found with the same ID that vary only by the case of the characters, Simulator will attempt to create a unique ID for each load. An appropriate message will appear in the message log.

Appending

- Appended data completely overwrites an existing record corresponding to the same bus number(s).
- Branch elements are appended to a case only if both of their terminal buses exist.
- Any new buses or branches that are created are now flagged as having just been added. This flag is then used as part of the pre-processing feature in the power flow solution to automatically estimate appropriate voltage magnitudes and angles for any existing or new branches and buses that become newly energized.
- A field, **EPC File\EPC Modification Status**, is populated to indicate whether an object is new or modified while reading.
- If a new three-winding transformer is entered which uses a bus number which is already used as a star bus by another three-winding transformer this will not be allowed and a new bus number will automatically be chosen.
- If appending an EPC file causes a transformer branch to be converted to a normal branch, a log warning message will show this.
- If the scheduled voltage for a bus is specified, the voltage setpoints for switched shunts and generators that are regulating this bus are also properly updated.

Saving

- When saving a case as a GE EPC file the Multi-Terminal DC converters will write out the "aloss" parameter in a manner consistent with the EPCL file mtTAP.p

- When using the [Present Topological Differences from Base Case](#) tool and the removed elements are saved in the GE EPC file format the elements are saved with a status value of -4.

Areva HDBExport Format (*.csv)

Reading

- When loading the hdbexport file, modified to automatically add two labels to each transmission line. One label starts with the "from" substation name and the other label starts with the "to" substation name.
- At the end of reading in an hdbexport CSV file, if any CBTyp values were encountered that were not recognized a dialog box will appear prompting the user to designate a Branch Device Type for each CBTyp (See the **hdbexport Files** section in [File Management](#) for more explanation on this).
- Additional bus labels are added when reading the MEAS records.

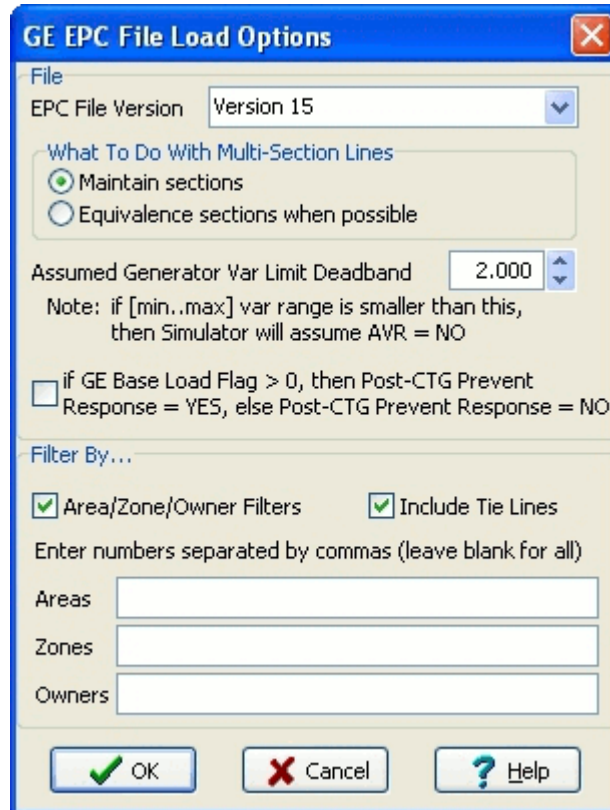
Auxiliary File Format (*.aux)

Simulator Auxiliary Files have now been greatly enhanced to include both Script sections for running Simulator commands in a "batch" process, and Data sections for changing mass amounts of load flow information. Many [auxilliary file formats](#) used in early versions of Simulator can not be saved in current versions. The older formats can still be read into Simulator.

Auxiliary file format and use now has its own help chapter titled [Auxiliary Script/Data Files](#).

GE EPC File Load Options

The GE EPC File Load Options dialog will open when choosing to open or save a GE EPC case "with options".



The screenshot shows the 'GE EPC File Load Options' dialog box. It has a blue title bar with a close button. The 'File' tab is selected. The 'EPC File Version' is set to 'Version 15'. Under 'What To Do With Multi-Section Lines', 'Maintain sections' is selected. The 'Assumed Generator Var Limit Deadband' is set to '2.000'. A note states: 'Note: if [min..max] var range is smaller than this, then Simulator will assume AVR = NO'. There is a checkbox for 'if GE Base Load Flag > 0, then Post-CTG Prevent Response = YES, else Post-CTG Prevent Response = NO'. The 'Filter By...' section has checkboxes for 'Area/Zone/Owner Filters' and 'Include Tie Lines', both of which are checked. Below these are three text boxes for 'Areas', 'Zones', and 'Owners'. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

EPC File Version

Set this to determine what version of the EPC file is being read or saved.

What To Do With Multi-Section Lines

Assumed Generator Var Limit Deadband

GE Base Load Flag Setting

When opening a file, the GE Base Load Flag setting will be used to set the Post-CTG Prevent AGC Response field in Simulator. The caption will be the following: **if GE Base Load Flag > 0, then Post-CTG Prevent Response = YES, else Post-CTG Prevent Response = NO**

When saving a file, the Post-CTG Prevent AGC Response field in Simulator will be used to set the Base Load Flag setting in the EPC file. The caption will be the following: **if Post-CTG Prevent Response = YES, then set GE Base Load Flag = 1, else set GE Base Load Flag = 0**

Filter By

If desired, only data in with specific areas, zones or owners can be loaded from the EPC file by using this option. Checking the Area/Zone/Owner Filters box will display the edit boxes for entering the Area, Zone, and Owner numbers. You can also check the Include Tie Lines box to read in branches for which only one of the two terminal buses is designated with the specified area, zone or owner.

Leaving this box unchecked will force the requirement that BOTH branch terminal buses meet the specified areas, zones or owners for it to be read from the EPC file.

See the [Case Formats](#) topic for information on special handling of data that occurs when loading a case using the GE EPC file format.

PTI RAW File Load Options

This dialog contains options related how Simulator reads (or writes) the PTI RAW file. There are only three options on the dialog.

RAW File Version

This option is available for both reading and writing the RAW file. Simply choose the appropriate version number from the drop-down menu.

Add comments for object labels at the end of RAW file records

This option is available for both reading and writing the RAW file. Simply check to append object labels as comments to the end of data records.

Creation of transactions for loads in a different area than their terminal bus

This is only relevant when loading a RAW file. The option allows the user to select if transactions should be created for loads that are in different areas than their terminal buses.

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) can be automatically created with the ID "RAW_LOAD". These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines only". If you know that your case was solved using "lines and loads", then you would not want to add the transactions. There are three options for how to deal with these possible interchange differences. The default is to let Simulator determine what is best based on the input data. If it is known what option was used in PTI, then the user can choose to either create the MW transactions or not create them. See the topic **Add and Remove transactions due to tie-line loads** with [File Management Options](#) for more information about adding or removing these transactions after the case has been loaded.

If not loading the RAW file with options, Simulator will determine based on the input data whether or not the transactions representing loads that are in different areas than their terminal buses need to be created.

When loading the RAW file with options, the user can select how to number the star buses of three-winding transformers. The options are to use a number that is near the primary bus number, start numbering above the maximum bus number, or specify a number at which to start the numbering.

When reading a RAW file that has comments on a given line, the comments will be loaded into the Memo field of the corresponding data record. However, the Memo field will not be written back out when saving a RAW file.

Three-Winding Transformer Star Bus Numbering.

The PTI RAW format does not include separate buses which represent the internal star buses of a three-winding transformer. When loading a RAW file, Simulator will automatically create buses for these star buses. This option provides the user a choice of how these star buses are numbered.

Opening a Simulation Case

The most common first step in using Simulator is to open a case. When a case is opened, any associated oneline files are also opened. To open a case, either select it from the [list of recently opened cases](#) in the file menu or

- Select **Open Case** from the [Application Button](#) to display the Open Dialog.
- In the **Type of Files** box, select the desired file type. By default, the PowerWorld Binary type is selected (*.PWB). The PowerWorld Binary is the preferred file type, providing the most comprehensive power system information along with the smallest size and quickest load time. Other file types include the PowerWorld Case type, PowerWorld Auxiliary file, PTI Raw Data formats, GE EPC text format, and IEEE Common Format. Please see [Case Formats](#) for more details.
- In the list of cases, click on the desired case.
- Click OK.

When opening an existing Simulator case, you may see one or more oneline diagrams pop up. When opening a power flow case created with another program, a oneline may not be available. However, you can easily create a oneline for such a case using the [Edit Mode](#).

If you are in the Edit Mode, you can modify the case. See [Edit Mode Overview](#) for details.

If you are in the Run Mode, the Clock Window may be displayed. This window shows the simulation start time, end time and current time. You may choose to close or open the clock at any time.

If the case has validation errors, the mode is immediately switched to Edit Mode. You must correct the validation errors shown in the message log before you can use the Run Mode to solve the case.

Opening a Online Diagram

Simulator supports online diagrams that have been developed using its Edit Mode tools. PowerWorld can also import PTI Draw Files (*.drw).

When a case is opened, any associated onelines are usually opened, as well. Additionally, online diagrams can be opened directly from any existing onelines using [online links](#). However, you may also directly open a online diagram using the following procedure:

- Select **Open Online** from the [Application Button](#).
- Choose the file type you would like to open by using the drop-down in the bottom right-hand corner of the Open dialog. The **Available File Type Options for Opening a Online** section details how the different file types work.
 - Alternately, you can use the file menu choices to open a online using a particular linking. The file menu choices are available on the submenu when selecting **Open Online** from the Application Button. The linking choices listed in the **Available File Type Options for Opening a Online** section found below are available with this option (except for the PTI Draw File format). When you select a particular linking option, you will be prompted if you want to save this as your default type. You can override the default by selecting a different linking option the next time that you open a online.
- In the list of available online files, click on the desired online.
- Click OK.

Available File Type Options for Opening A Online

- **Online Display File (*.pwd)** - This is the standard binary file used to store online diagrams. When this option is selected, online display objects involving buses will be linked to their corresponding case data objects through bus numbers.
- **Online Display File (Name_kV linking) (*.pwd)** - This is the standard binary file used to store online diagrams, but a special table stored with the file is used to link the online display objects involving buses to their corresponding case data objects using bus name and nominal kV combinations instead of bus numbers and renumber the bus numbers of the display objects to match the current case. This option is useful when opening a online that was created with a different case than the one currently in memory and the current case has undergone some bus renumbering, but the bus names and nominal kV levels remain the same as in the original case.
- **Online Display File (Label linking) (*.pwd)** - This is the standard binary file to store online diagrams, but a special table stored with the file is used to link the online display objects by [label](#) to their corresponding case data objects. Only objects that allow the use of labels will be linked in this manner. If a linking via label cannot be found, the online object will be unlinked. This option is useful when opening a online that was created with a different case than the one currently in memory and the current case has undergone bus renumbering, but the labels remain the same as in the original case.
- **PTI Draw File (*.drw)** - File format supported by PSS/E. All functionality of that file format may not be fully supported in Simulator.
- [Display Auxiliary File \(*.axd\)](#) - This is a text file used to store the online display objects.

You may open as many onelines as you like, and even multiple copies of the same online.

The following topics may be helpful when opening onelines created with a different case:

[Refresh Anchors](#)

[Bus Renumbering Dialog](#)

Three-Winding Transformer Star Bus Automatic Renumbering

When opening a oneline diagram by using one of the PWD options, the star buses of three-winding transformers will automatically be renumbered to match the current power flow case. This is done without any input from the user. This is done because some power flow case file formats do not require that star bus numbers remain constant, therefore, if opening a oneline diagram with a power flow case other than the one with which it was created, there is a good possibility that the star bus numbers will be different. The renumbering is accomplished using a table saved with the PWD file that contains the three-winding transformers. If using the name_kV linking to open a oneline, the three-winding transformers in the internal table are identified by name_kV when doing the star bus updates. If the transformer cannot be found when using either the number or name_kV linking, its star bus will not be renumbered.

Recently Opened Cases

A numbered list of the most recently opened cases appears in the Recent Files list when you click on the [Application Button](#). Simply choose a case from the list to open the case.

Building a New Case

To create a new case in Simulator, select **New Case** from the [application file menu](#). After asking you whether or not to save the current working case (if one exists), Simulator will automatically switch to Edit Mode. The screen will turn to the default background color, indicating that you can begin to build the new case.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance.

Building a New Online

This option enables you to create a new online diagram. It is accessible from either the Edit or Run modes by choosing **New Online** from the [application file menu](#). When you select it, the application will automatically switch to Edit Mode, from which you can construct the new online.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance. Additionally, the following topics may be of some assistance:

[Using the Insert Palettes](#)

[Online Conditional Display of Objects](#)

[Using the Online Alignment Grid](#)

Saving Cases

Simulator allows users to save case information in several different formats. To call up the Save As dialog, select **Save As** from the [Application Button](#). The format in which the file will be saved depends upon the value of the **Save as Type** field of the dialog. Simulator can save cases in PowerWorld binary format (the default), PowerWorld Auxiliary file format, PTI raw data, GE PSLF format, and the IEEE common format. Please see [Case Formats](#) for more details on the various case formats available.

To force saving a case with comments, a submenu exists for the **Save As** and **Save As Type** menu items. Select the **... with Comment...** to open the comment dialog. More information about Case Comments can be found in the [Case Description](#) topic.

To convert a case to a different format, follow this procedure:

- Select **Open Case** from the [Application Button](#).
- In the **Files of Type** box, select the desired file type. By default, the PowerWorld Binary type is selected (*.PWB).
- Select the name of the desired case and click *OK*.
- Select **Save Case As** from the [Application Button](#).
- Change the **Save as Type** setting to match the desired file type, designate the name with which to save the file, and click *OK*.

To simply save the case with its current name and format, select either **Save** from the [Application Button](#) or the **Save Case** button on the Quick Access toolbar.

Saving a Oonline

Select **Save Oonline** from the [Application Button](#) to save the currently selected oneline file.

Select **Save Oonline As** from the [Application Button](#) to save the currently selected oneline file with a different name or format. You may overwrite existing files.

For both the Save Oonline and Save Oonline As options, only the oneline diagram is saved; the case (i.e. the power flow model) is not written to disk.

Close Online

To close a online diagram, select to **Close Online** from the [Application Button](#), or right-click on the background of the online diagram and select **Form Control > Close** from the local menu. This will prompt you to save the online if changes have been made since the last save. Note that this does not close the current case, just the online diagram. The power system data is still loaded, and case simulations can be run with the online closed.

Generator Capability Curves Format (*.gcp)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator capability curves are used to model the dependence of the generator reactive power limits and the generator's real power output. The generator reactive capability curves files have the following format, with one generator per line:

num or name, ID, P₁, Q_{1,max}, Q_{1,min}, P₂, Q_{2,max}, Q_{2,min} ... P_n, Q_{n,max}, Q_{n,min}

where

num/name the generator's bus number or the bus' name in single quotes,

ID the generator's single character id,

P_i a generator MW output value,

Q_{i,max} and Q_{i,min} the associated maximum and minimum reactive power limits.

Version 11 and later of Simulator allow for an unlimited number of capability curve points to be read from the gcp file. Up to 10 different P/Q_{max}/Q_{min} values may be specified in older versions of Simulator.

Generator Cost Data Format (*.gcd)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator cost data file contains the parameters used to model the generator operating costs. Each record in the generator cost data file specifies the operating cost model for a single generator. The cost model may be either a cubic polynomial or a piecewise linear curve. When reading a file of generator cost data records, Simulator will open the **Cost Curve Data Options Dialog** below. This dialog gives you additional control over the input and process of generator cost information.

Each generator cost data record begins with the fields

num or name, ID,

where

num/name the generator's bus number or the bus' name in single quotes,

ID the generator's single character id,

Following the ID field is an optional single-character field **CostModel** that specifies the type of cost model. If **CostModel** is 'CUBIC' or is not specified, the record describes a cubic cost model and thus concludes with the following fields:

a_i, b_i, c_i, d_i, FuelCost ParFac

where

a_i, b_i, c_i, d_i input-output curve coefficients; see [Generator Cost Information](#) for details,

fuelcost fuel cost, expressed in price per Mbtu,

ParFac generator participation factor.

If, on the other hand, **CostModel** is 'PLIN', the record describes a piecewise linear cost model. The remainder of the record specifies pairs of generator output and corresponding generator cost:

FixedCost MW₁ IncCost₁ MW₂ IncCost₂ ... MW_n IncCost_n MWneg ParFac

where

FixedCost Operation cost independent of the generator's MW output; expressed in \$/Hr,

MW_i, IncCost_i Output/incremental cost data point pairs that define the incremental cost model for the generator (note IncCost_i has the units \$/MWhr),

MWneg An arbitrary negative value used to terminate the record,

ParFac generator participation factor.

Cost Curve Data Options Dialog

The **Cost Curve Data Options Dialog** gives you additional control over the input and processing of generator cost information. It appears when you try to read generator cost information from an [auxiliary data file](#) (either [.aux](#) or [.gcd](#)).

To command Simulator to turn all units onto automatic generation control (AGC) for which it reads a cost data record, check the first checkbox. This box has the rather lengthy label "If a unit for which a cost curve is read is not on AGC, Simulator should set it on AGC." In other words, if a cost curve is read for a particular generator, that generator will be set on AGC if this box is checked.

Cross-compound is a generator architecture sporting a single boiler and two turbines, one operating at high pressure and the other operating at low pressure. The existence of cross-compound units may complicate the modeling of generator cost characteristics, because sources for this information tend to describe the cost associated with the tandem, whereas Simulator's economic dispatch will try to dispatch the two parts of the cross-compound unit as two separate units. To address this problem, Simulator gives you the option to lump the properties of the high- and low-pressure turbines into a single unit. It consolidates the two components by adding together their maximum MW outputs, minimum MW outputs, and present output levels, assigning these quantities to one of the units, and setting the other unit off AGC and at 0 MW of output. To have Simulator perform this function for you, check the box in the *Cross Compound Units* panel.

To tell Simulator how to identify units that belong to the same cross-compound generator, click the box labeled "How To Identify...". This will open the **How to Identify Cross-Compound Dialog** described next.

How to Identify Cross-Compound Dialog

Simulator regards cross-compound units as those that are connected to the same bus and that have generator id's that match any of a number of specified pairs. You must define for Simulator the pairs of generator id's that should be used to identify cross-compound units. Use the "**How to identify cross-compound units**" dialog to do this.

Specify the pair of generator id's in the small text boxes below the text labeled "Specify a new pair of id's that identify two units that belong to the same cross-compound set." Then, click the **Add** button to include the pair of id's you just entered in Simulator's cross-compound identification procedure. The box on the right lists all the pairs of identifiers Simulator will use to identify parts of a cross-compound set. To remove a pair of identifiers from this list, select it from the box and click the Delete key.

The list of cross-compound identifier pairs is stored in the pwrworld.ini file, so there is no need to specify these pairs each time you load in a new case.

Injection Groups Format (*.inj)

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The injection group files store the information for defined injection groups (also referred to as participation groups). The files have the following format:

GROUP groupname

POINTS

Devtype busnum id participation pointtype

·
·
·

END

where

groupname the name of the injection group, in single quotes,

devtype either GEN or LOAD,

busnum bus number of the device,

id the device's single character ID,

participation relative amount each device will contribute during a transfer,

pointtype either FIXED or DYNAMIC.

Multiple GROUP sections can be put in the file, each marked with END to signify the end of that group's data.

Interface Data Format (*.inf)

These files are outdated for Simulator version 8 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The interface data files store the interface data. The files have the following format:

INTERFACE intname lima limb limc
Elemtype from to id checktoend fromto

·
·
·

where

intname the interface name, in single quotes,

lima, limb, limc the three limits for the interface, in MW,

elemtype one of three valid element types: AREA, LINE, or ZONE

from for AREA and ZONE, the first area or zone number; for LINE, the from bus number,

to for AREA and ZONE, the second area or zone number; for LINE, the to bus number,

id only included for LINE, the two character circuit identifier; otherwise left blank,

checktoend only included for LINE, if this field equals 1, the flow is checked at the to bus, otherwise the flow is checked at the from bus; otherwise left blank,

fromto only included for LINE, if this field equals 1, the positive flow is assumed to be from the from bus to the to bus, otherwise the flow is assumed to be in the opposite direction; otherwise left blank.

Multiple interface records can be listed in the file, and multiple interface elements can be listed in each interface record.

Sequence Data Format

The sequence data files store the sequence data needed to run a fault analysis. The sequence data format has the following five different types of records:

- Generator
- Load
- Branch
- Switched Shunt
- Mutual Impedance

Each of these four types of sequence data records has a different format, since they each store different information for the sequence data. The format of each record is given below:

Generator

GEN busnum ID Rpos Xpos Rneg Xneg Rzer Xzer RN XN

where

busnum generator terminal bus,
id generator identifier,
Rpos, Xpos positive sequence resistance and reactance,
Rneg, Xneg negative sequence resistance and reactance,
Rzer, Xzer zero sequence resistance and reactance,
RN, XN neutral-to-ground resistance and reactance.

Load

LOAD busnum Gneg Bneg Gzer Bzer

where

busnum load terminal bus,
Gneg, Bneg negative sequence total conductance and susceptance for all loads at bus,
Gzer, Bzer zero sequence total conductance and susceptance for all loads at bus.

Branch

BRANCH fbusnum tbusnum ckt Rzer Xzer Czer fGzer fBzer tGzer tBzer Xftype

where

fbusnum, tbusnum from and to bus numbers for the branch,
ckt branch circuit identifier,
Rzer, Xzer zero sequence branch resistance and reactance,
Czer total zero sequence line charging,
fGzer, fBzer zero sequence line shunt conductance and susceptance at the from bus end of the branch,

tGzer, tBzer zero sequence line shunt conductance and susceptance at the to bus end of the branch,

Xftype Transformer configuration, entered as an integer based on the following table. Note that the default for a transmission line is 3, since a grounded wye - grounded wye transformer connection has the same equivalent model as a transmission line.

- 0 Wye - Wye
- 1 Grounded Wye - Wye
- 2 Wye - Grounded Wye
- 3 Grounded Wye - Grounded Wye
- 4 Wye - Delta
- 5 Delta - Wye
- 6 Grounded Wye - Delta
- 7 Delta - Grounded Wye
- 8 Delta - Delta

Switched Shunt

SSHUNT busnum numblocks Bzer1 Bzer2 ...

where

busnum terminal bus number,

numblocks number of different zero sequence admittance blocks,

Bzer# zero sequence susceptance for each admittance block, maximum of 8.

Mutual Impedance

MUTIMP from1 to1 ckt1 from2 to2 ckt2 RM XM start1 end1 start2 end2

where

from1, to1 from and to bus numbers of the first mutually coupled branch,

ckt1 circuit identifier of the first mutually coupled branch,

from2, to2 from and to bus numbers of the second mutually coupled branch,

ckt2 circuit identifier of the second mutually coupled branch,

RM, XM zero sequence mutual resistance and reactance,

start1, end1 start and end locations of the section of the first line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1),

start2, end2 start and end locations of the section of the second line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1).

Exporting Onelines in Different Graphic Formats

Simulator can export oneline diagrams and other graphical displays as bitmaps, metafiles, jpegs, or gifs. Select **Export Oneline** from the [Application Button](#). A save file dialog appears where the dropdown menu for files type offers four choices: JPEG, BITMAP, METAFILE, GIF. Select the file type and choose the name before clicking the save button.

Exporting an image as a jpeg also requires you to set the compression ratio for the picture. See [Saving Images as Jpegs](#) for more information.

Exporting an image as a gif provides the option to save an animated image. Upon saving, you are given the option to specify the number of frames to save and the time delay between frame changes during playback. You are also given the ability to scale the image before saving.

Saving Images as Jpegs

Simulator can save oneline diagrams, bus view displays, and strip charts as jpeg images. To save a oneline diagram or bus view display as a jpeg, select **Export Oneline** from the [Application Button](#). This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. This brings up the Jpeg Options Dialog where you decide the picture's resolution. Adjust the resolution control to specify the compression ratio at which to save the diagram as a jpeg. The greater the resolution you specify, the larger the resulting file will be. Click **Save** to save the image or click **Cancel** to terminate the process without saving the image as a jpeg.

To save a strip chart as a jpeg image, right-click on the background of the strip chart and select **Export Image** from the resulting local menu. This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. Specify the compression/resolution in the Jpeg Options Dialog. Click **Cancel** to terminate the process, or click **Save** to save the jpeg.

Saving Admittance Matrix and Jacobian Information

The **Save Ybus or Power Flow Jacobian Dialog** is used to store the power system bus admittance matrix (Ybus) and/or the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®. This dialog is primarily designed for users doing power system analysis research. The dialog has the following fields.

Ybus in MATLAB Format

Filename for Saving Ybus

Enter the name of the file in which to store the Ybus data. The Ybus data is stored using the MATLAB sparse matrix format in the matrix **Ybus**. If the **Include Bus Voltages** field is checked, then the bus voltages are also stored, but in the vector **V**.

Save Ybus in MATLAB Format

Click this button to save the Ybus.

Power Flow Jacobian in MATLAB Format

Jacobian Save File

Enter the name of the file in which to store Jacobian data. The Jacobian is stored using the MATLAB sparse matrix format in the matrix **Jac**.

Jacobian ID Save File

Enter the filename to store the text identifier information. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

File Type

Choose the type of MATLAB file you wish to save as. The **MATLAB .M Format** is the more common text format used for directly loading MATLAB information. The **Text for MATLAB Ascii** is for use with MATLAB's ability to read Ascii files. The Ascii file type can be read into MATLAB much faster than the traditional .M files.

Jacobian Form

Select **Rectangular** to store the rectangular form of the Jacobian, or **Polar** to store the polar form of the Jacobian.

Save Jacobian

Click this button to save the Jacobian and object identifier information.

Working With GE EPC Files

PowerWorld Simulator can open and export to GE EPC files, but as with any format conversion there are some quirks to keep in mind. Many of these are described in a number of pages throughout these help files.

- [Differences in File Formats](#)
- [Appending a Case](#)
- [Case Formats](#)
- [File Management Options](#)
- [GE EPC File Load Options](#)
- [Present Topological Differences from Base Case](#)
- [Script General Actions](#)
- [Transient Stability Data from External Files](#)
- [Transient Stability Overview: Generator Models](#)

Differences In File Formats

PowerWorld Simulator supports loading from and saving to a number of different external formats, including EPC and PSS/E files. There are a number of points where Simulator's handling of a power system is not precisely represented in other formats, so some care must be taken when working with these other files.

GE EPC Files

Buses

Bus names are limited to 12 characters. When writing out to PSLF files, Simulator will truncate names in a way that ensures unique names remain unique.

PSLF specifies the specific voltage setpoint with a Bus record, and then specifies a range with the device (Vband for a shunt, Vmin/Vmax for a transformer)

Generators

GE Baseload Flag is an extra flag used by PSLF in transient simulation only (user-written EPCL code might also use this). Simulator allows you to interpret this field as a Post-Contingency AGC status (Simulator has a separate field for Transient Simulation use).

PSLF supports up to 8 owners per generator, while Simulator only supports 4. If an EPC file specifying more than 4 owners is loaded, Simulator will keep track of them with EPC specific fields.

Simulator has an AVR flag which is set to YES or NO. PSLF uses (Qmax-Qmin) to determine AVR. By default ranges at 2.0 Mvar or less are considered AVR=NO.

Shunts

PSLF supports up to 4 owners per shunt, but Simulator assumes ownership is inherited from the terminal bus. Simulator will keep track of any owners specified in a loaded EPC file with EPC specific fields.

	GE PSLF	PTI PSS/E	PowerWorld Simulator
Controllable Shunt	SVD (Multiple allowed per bus)	Switched Shunt (Only one allowed per bus)	Switched Shunt Control Mode = Fixed, Discrete, or Continuous Multiple allowed per bus (coordinated in version 16)
Fixed Bus Shunt	SHUNT (Multiple allowed per bus)	Fixed Bus Shunt (multiple allowed per bus)	Switched Shunt Control Mode = Bus Shunt (multiple allowed per bus)
Fixed Line Shunt	SHUNT Assigned to a line. IDs must start with an <i>f</i> or <i>t</i>	No special record. Fields with AC Line <i>GI, BI, GJ, BJ</i> specify the total at each end	Line Shunt Each end of a line can have multiple shunts. IDs assigned for multiple shunts.
G and B for equiv	None	None	Bus fields <i>Nom G Shunt</i> and <i>Nom B Shunt</i>

Branches

PSLF supports up to 8 owners per branch, while Simulator only supports 4. If an EPC file specifying more than 4 owners is loaded, Simulator will keep track of them with EPC specific fields.

Transformers

PSLF allows a tertiary bus number of zero (0). Simulator supports this exclusion of the tertiary bus, but will write it as two 2-winding transformers when exporting to a RAW file.

Multi-Section Lines

In PSLF, intermediate buses are not defined and thus loads, generators, and shunts at these points are not allowed. When writing out to an EPC file, the multi-section line record is ignored if intermediate buses have objects.

Area/Zones for Shunt Objects

PSLF allows area and zone designations for specific branches and transformers. Simulator reads this into EPC specific fields, but does not use them for anything.

Area MW Transactions

PSLF includes tables showing Transactions. The actual scheduled values are informational only. PSLF does not use these MW Transactions in the power flow solution; instead, it includes a field with an area specifying the net MW export. When loading an EPC file, Simulator's area field "Unspecified MW Interchange" is assigned as the "leftover" which isn't included in the transactions table.

DC Lines

In Simulator, there are distinct container objects for both a two-terminal DC line and a multi-terminal DC line (MTDC). In PSLF, there are only definitions of the DC Buses, AC/DC converter, and DC line objects. The groupings are then determined by the topology of the DC line links. PSLF requires that all DC buses throughout the entire model have a unique DC bus number, which Simulator supports.

Interfaces

PSLF supports the definition of an interface as a group of AC lines (or Multi-Sections lines). Simulator supports several more types of interface elements, such as Area-To-Area, Zone-To-Zone, DC Line, Injection group, Generator, Load, or another interface.

Injection Groups

Simulator supports the definition of an injection group which represents a group of generators, loads, or switched shunts (and can contain other injection groups). This can be used for summary calculations, sensitivity-based calculations, or as a source and sink for a transfer. A user-written EPCL would be required to support this in PSLF.

Substations

In Simulator, each bus can be optionally assigned to a substation. This is useful in oneline diagram creation and is necessary to support EMS-based full-topology models. PSLF does not support substations.

Super Areas

In Simulator, Super Area objects can be created and each area can be assigned to one super area. This is useful in studying groups of areas which are dispatched together (Electricity Markets). PSLF does not support Super Areas.

Nomograms

In Simulator, Nomogram objects can be created which specify two interfaces and then a two-dimensional limit region. Nomogram limit violation can then be monitored or enforced in Contingency Analysis, ATC, OPF and SCOPF studies. PSLF does not support Nomograms.

PSS/E Files

Buses

PSS/E bus names are limited to 12 characters. When writing out to PSS/E files, Simulator will truncate names in a way that ensures unique names remain unique.

Generators

While Simulator specifies the capability curve explicitly, PSS/E has a separate input file called "GCAP" which specifies the capability (Simulator can read this file).

PSS/E specifies a "bus type" to signify the generator is not on AVR control, while Simulator uses an AVR flag set to YES or NO.

Shunts

See shunt format comparison table above.

Transformers

PSS/E does not support a fixed tap definition on the same side as the variable tap (FROM), thus Fixed tap is always assumed as 1.00.

PSS/E does not specify the intermediate Star Bus of a 3-winding transformer explicitly. The voltage and angle at this point are part of the transformer record.

PSS/E does not allow the tertiary of a 3-winding transformer to be excluded. Simulator allows this, though we do not consider it good practice.

Area/Zones for Shunt Objects

Simulator allows Generators, Loads and Switched Shunts to have a different area or zone than the terminal bus. PSS/E only allows this for Loads. These objects are then treated as tie-lines in area and zone interchange calculations. PSS/E has a specoal solition option to specify whether to include the interchnge as "LINES ONLY" or "LINES AND LOADS". Simulator always uses Lines and Loads.

Area MW Transactions

PSS/E includes tables showing Transactions. The actual scheduled values are informational only. PSS/E does not use these MW Transactions in the power flow solution; instead, it includes a field with an area specifying the net MW export. When loading an EPC file, Simulator's area field "Unspecified MW Interchange" is assigned as the "leftover" which isn't included in the transactions table.

DC Lines

PSS/E requires that all DC buses of a particular MTDC grouping be numbered from 1 to the number of DC buses. Simulator supports this when exporting to PSS/E.

Interfaces

PSS/E has a monitor functionality (MON file) which allows definition of an interface which includes AC branches and Line Open of Close contingent elements.

Injection Groups

Simulator supports the definition of an injection group which represents a group of generators, loads, or switched shunts (and can contain other injection groups). This can bus used for summary calculations, sensitivity-based calculations, or as a source and sink for a transfer. PSS/E has a similar concept called a "subsystem", which you may find in a SUB file.

Substations

In Simulator, each bus can be optionally assigned to a substation. This is useful in oneline diagram creation and is necessary to support EMS-based full-topology models. PSS/E does not support substations.

Super Areas

In Simulator, Super Area objects can be created and each area can be assigned to one super area. This is useful in studying groups of areas which are dispatched together (Electricity Markets). PSS/E does not support Super Areas.

Nomograms

In Simulator, Nomogram objects can be created which specify two interfaces and then a two-dimensional limit region. Nomogram limit violation can then be monitored or enforced in Contingency Analysis, ATC, OPF and SCOPF studies. PSS/E does not support Nomograms.

Overview of PowerWorld Simulator Project Files

Performing a simulation using PowerWorld Simulator sometimes requires using a number of different files. In addition to the case file that stores the model of the system, there may be one or more oneline diagrams depicting various regions of the system. These oneline diagrams might feature [document links](#) that connect to files that were created using other applications, as well as [oneline links](#), which open other oneline diagrams when they are clicked. For these links to function properly, the documents and oneline displays to which they connect must be available. A simulation might also employ a template file that is used to load in a predefined set of solution, environment, and display options. Furthermore, the simulation might make use of data stored in [auxiliary data files](#) to supplement the data stored in the case file. Finally, a simulation might utilize a [script file](#) to perform some automated sequence of tasks, or even to display a movie of system conditions. Having to deal with so many files may make it difficult to transfer the case to another computer, or to share the model with a colleague. PowerWorld Simulator Project files provide a solution.

PowerWorld Simulator Projects have the extension *.pwp. A project serves as a container for all the files that might comprise a simulation, including the case file, one or more oneline diagrams, a script file, one or more auxiliary data files, and a case template, as well as any other support files you may wish to include. A project is actually a compressed file archive that is compatible with the widely available PKZip and WinZip file compression utilities. It is strongly recommended, however, that you work with project files strictly within the PowerWorld Simulator environment, as Simulator can automatically perform the file compression and extraction functions and process the contents of the included files in a single step. (Note: Simulator performs file compression and extraction using software components available from <http://www.cdrom.com/pub/infozip/> and <http://www.geocities.com/SiliconValley/Orchard/8607/main.html>).

PowerWorld Project Initialization Script

Included in every project file is an initialization script file called OPENPWB.SCP. This is a special type of PowerWorld script file that is used to unload the contents of the project that are used in the simulation. This file must begin with the keyword "INITIALIZATION" and terminate with the keyword "END". Between these two lines, OPENPWB.SCP identifies the name of the case, any oneline diagrams that should be opened immediately, a template file (if any), a script file (if any), and one or more auxiliary data files (if any) to open and read into memory after extracting the files. The file might also contain the keyword "AUTOSTART", indicating that the simulation of the system described by the project file should commence immediately after it is read into memory. The OPENPWB.SCP is generated for you automatically when you save a new project file. Here is an example that loads the case B7FLAT.PWB (in PowerWorld binary format) along with oneline diagram B7FLAT.PWD, the template B7FLAT.PWT, the script B7FLAT.SCP, and the auxiliary file B7FLAT.AUX, and then starts the simulation immediately after all contents are extracted and read:

```
INITIALIZATION
CASE PWB B7FLAT.PWB
ONELINE B7FLAT.PWD
TEMPLATE B7FLAT.PWT
SCRIPT B7FLAT.SCP
DATAFILE B7FLAT.AUX
AUTOSTART
END
```

Again, all projects must contain the file OPENPWB.SCP. If a project does not contain this file, an error message will be shown. Simulator automatically includes an OPENPWB.SCP file with every project it creates. Therefore, unless you try to create a project file outside of Simulator, you will not have to worry about this requirement.

Associating Project Files with Simulator

PowerWorld Simulator can automatically open and load the contents of a project if the proper file associations have been made. When Simulator is installed on a computer, it registers the file extension *.pwp in the Windows registry. Then, whenever you double-click on an icon for a project file or download a project file from a web site, Simulator will start, extract the contents of the project file, and read them into memory. The power system will then be displayed exactly as the author of the project file intended it to be displayed.

If the file association is somehow destroyed (i.e. Windows no longer recognizes that *.pwp files should be associated with and opened by Simulator), you can re-establish the association manually using the following procedure:

- Windows 95/98/NT
 - o Open Windows Explorer.
 - o Select View > Folder Options from Windows Explorer's main menu.
 - o Switch to the tab labeled "File Types."
 - o Click the button labeled "New Type."
 - o For "Description of Type," specify "PowerWorld Project."
 - o For "Associated Extension," specify "PWP".
 - o For "ContentType (MIME)," specify "application/powerworld_project."
 - o For "Default Extension," specify ".PWP".
 - o Under "Actions," click the button labeled "New."
 - o Specify the "Action" as "open."
 - o Specify the "Application used to perform action" as the path to the Simulator executable file, followed by a space and the string "%1" (including the quotation marks). For example, if the path of the PowerWorld Simulator executable is "c:\program files\PowerWorld\pwrworld.exe," then you should specify the following string:
c:\program files\PowerWorld\pwrworld.exe "%1"
 - o Click OK to close the Action dialog.
 - o Click OK to complete the definition of the file association.
- Windows XP
 - o Open Windows Explorer
 - o Select Tools > Folder Options from Windows Explorer's main menu.
 - o Switch to the tab labeled "File Types"
 - o Click the button labeled New
 - o For "File Extension", enter "PWP", then click OK.
 - o Find the PWP entry in the Registered file types list, then click on it.
 - o Next, with the PWP entry selected, click on Advanced.
 - o In the box to the left of the "Change Icon..." button, type in "PowerWorld Project"
 - o Click on the "New..." button on the right-hand side of this dialog, which will pop up a "New Action" dialog
 - o In the New Action dialog, click "Browse..."
 - o Navigate to and select the PowerWorld program binary (pwrworld.exe); the default location is c:\Program Files\PowerWorld\pwrworld.exe
 - o In the New Action dialog, enter "Open the project" in the field labeled "Action"
 - o Click OK in the New Action dialog and in the Edit File Type dialog
 - o Click Close when you are back to the main File Types dialog.
- For future Windows versions, refer to the Windows help on changing file type associations.

The *.pwp file type will now be recognized on your system as being associated with PowerWorld Simulator. Again, you will need to perform this procedure only if the association somehow gets removed.

Creating a New Project File

Creating a new project file in Simulator is fairly simple. First, make sure that the case for which you want to create the project is open in Simulator, as well as any oneline diagrams that you wish to open automatically whenever the user opens the project. Then select **Save As Project** from the [Application Button](#). This opens the [Create Project Dialog](#), which you can use to specify the contents of the project file. When you click OK on the Create Project Dialog, the files you identified will be compressed into a single file, along with the [project initialization script](#) that tells Simulator how it should import the contents of the project.

Create Project Dialog

The Create Project Dialog enables you to create a new [PowerWorld Simulator Project](#) by specifying the files that should be included in the project. To open the Create Project Dialog, select **Save As Project** from the [Application Button](#). When it opens, the dialog will automatically include the case file, all open oneline diagrams, all documents and onelines linked to the display by [document links](#) and [oneline links](#), the current case template file, and the current script in its list of files to add to the project. You can then use this dialog to add files to or delete files from the list of project contents.

The Create Project Dialog features the following controls:

Case

Identifies the name of the case file that is currently open in Simulator. The case file may be in PowerWorld binary format, in PTI Version 23 - 29 .raw format, or in GE PSLF .epc format. You cannot change this field, because it must be set to whatever case is currently open.

Open these oneline diagrams automatically

Lists the names of the oneline diagrams that will automatically open whenever a user opens the project file. This is also a read-only control and lists the names of all oneline diagrams that are currently open in Simulator. Therefore, before you start to create a project using the Create Project dialog, make sure that only those oneline diagrams you wish to open automatically with the project are currently open in Simulator.

Use this script file

Identifies a [script file](#) that should be opened automatically when the project is open. Scripts are text files that contain commands in [PowerWorld's scripting language](#) that Simulator interprets to perform a predefined set of tasks during the simulation. If a script file is in use when you try to create the project, its name will automatically appear in this text box. To specify a different script file, either type its full path in the text box, or press the adjacent *Browse* button to search for it on your machine.

Load data from auxiliary files

Lists the [auxiliary data files](#) that should be automatically read when the project is open. Auxiliary data files are text files that contain data that supplement or supersede the information in the case file. You may include as many automatically loading auxiliary data files in the project as you like. To add an auxiliary file to the project contents, click the adjacent Add button and locate the file on your machine. To remove an auxiliary file from the list, select it and click the adjacent Remove button.

Also include these files

Lists any other files you may wish to include in the project. Use this control to include supplementary documents, pictures, or even other PowerWorld project files, in the project. To add a file to the contents, click Add a File. To remove a file from the list, select it and press the adjacent Remove button.

Automatically start the simulation after opening the project

Check this button to cause Simulator to start the simulation immediately after opening the project. This saves the user who opens the project the step of switching to run mode and clicking the play button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. This option is particularly effective, for example, if you want to display a "movie" of system conditions when the project is open. By including a script file in the project that tells Simulator how to modify the system and view over time, you create a project that, when opened or downloaded, animates the scenario you are trying to model.

Save this project as

Identifies the name of the project that should be saved. By default, the project name is the same as that of the case except that it has the extension "pwp". To specify a different name, either type it directly in the text box, or click the adjacent Browse button and specify the full path.

When you have finished specifying the contents of the project file, click the **OK,save the project** button. If you wish to cancel the operation without creating the project, click **Cancel**.

Opening an Existing Project

To open an existing [PowerWorld Simulator Project file](#), select **Open Project** from the [Application Button](#), and select the name of the project you wish to open. Alternatively, you may select **Open Case** from the [Application Button](#), change the *Files of Type* setting to "PowerWorld Project (*.pwp)," and select the project you wish to open. Simulator will extract the contents of the project file into the project file's folder and input the information it needs to display the case, oneline diagrams, and other associated files.

Model Explorer



Most [Case Information Displays](#) are enclosed inside the Model Explorer. To open the Model Explorer, click on the Model Explorer button shown above. The Model Explorer button is available on both the [Case Information Ribbon Tab](#) and [Tools Ribbon Tab](#) by default. It is also placed on the [Quick Access Toolbar](#) by default.

Note: normally only one Model Explorer window will be open at a time. If you choose to open a case information display from various places in the Ribbon, the open Model Explorer will be moved to this case information display. If you want multiple Model Explorers to be open, then click on the **Open New Explorer** button below the Explore Pane.

When you open the Model Explorer you will see a case information display. In addition to this the model explorer has 5 additional regions to help you navigate the model as well as modify the model.

Explore Pane

The Explore Pane contains a hierarchical list of most of the objects contained in the power system model. Entries in the Explore Pane that are grayed out represent objects that presently are not defined in the model. The Explore Pane is separated into eight primary folders, with only the Network and Aggregations folder expanded by default. The primary folders are as follows:

- **Recent** - contains a list of case information displays that have been recently opened. This same list of case information displays are depicted by the Tabs displayed across the top of the Model Explorer which indicate the currently open data tables.
- **Network** - contains a list of the network model objects. Generally these represent actual physical devices such as a generator or transmission line.
- **Aggregations** - contains a list of model objects that represent an aggregation of other model objects. Some examples of data found here are [interfaces](#) and [injection groups](#).
- **Solution Details** - provides access to case information displays that provide details regarding the power flow solution. It includes displays such as the Y-Bus, Jacobian, and summaries of remotely regulated buses
- **Case Information and Auxiliary** - contains generic objects that operate on the other objects in the model. This includes [filters](#), [expressions](#), and model criteria.
- **Contingency Analysis** - contains a convenient place to access case information displays showing options related to [contingency analysis](#).
- **Fault Analysis** - contains a convenient place to access case information displays showing options related to [fault analysis](#).
- **Optimal Power Flow** - contains a convenient place to access case information displays showing options related to the OPF or [SCOPE](#).
- **Transient Stability** - contains a convenient place to access case information displays showing options related to [Transient Stability](#)
- **User-Defined** - contains a list of User-Defined Case Information Displays available. These displays are created by right-clicking on the Explore Pane and choosing to Insert User-Defined Case Info. For more information see the [User-Defined Case Information Displays](#) help.


Explore Pane Options

There are a few options regarding the Explore Pane which can be accessed by right-clicking on the Explore Pane, Searchbar, Filterbar, or Case Information Toolbar. When right-clicking a local menu will appear that allows you to specify the following options.

Toolbar (Top)	▶	Fields Pane Location : Allows you to specify the location of the Fields Pane. See the Model Explorer: Fields Pane for more details.
Searchbar (Bottom)	▶	
Filterbar (Top)	▶	
Fields Pane Location	▶	Save Recent (Number) : Choose this option to change the number of recently viewed case information tables that are kept open and shown as tabs in the Recent Tabs.
Save Recent (10)		
Show Number of Objects		Show Number of Objects : Check this option to show a number next to each entry in the Explore Pane.
Insert User-Defined Case Info		
Remove User-Defined Case Info		

Insert User-Defined Case Info, Remove User-Defined Case Info : These options allow you to insert and remove user-defined Case Information Displays. For more detailed information see the [User-Defined Case Information Display](#) help.

Fields Pane

The Fields Pane provides direct access to modifying the columns that appear on the case information displays. This replicates the features that can also be accessed by choosing  **Display/Column Options** on the [case information toolbar](#) to open the dialog for [Configuring the Case Information Displays](#). See the [Model Explorer: Fields Pane](#) for more details.

Recent Tabs

The Recent Tabs represent a list of the most recently viewed case information displays on the Model Explorer. You can click on the X to the left of each tab to clear that from the recent list.

Case Information Toolbar

The Case Information Toolbar contains a graphical representation of all the options and actions which can be applied to the case information display. The toolbar is described in detail in the [case information toolbar](#) topic. Also note that all the options displayed on the toolbar are also available by right-clicking on the toolbar and showing the [case information local menu](#).

Case Information Filterbar

The Filterbar provides convenient access to apply an [Advanced Filter](#) or a [Device Filter](#) to the case information display. The toolbar is described in detail in the [Case Information Filterbar](#) topic.

Case Information Searchbar

The Searchbar contains a toolbar that allows you to search the active case information display for particular text. This toolbar behaves in the same manner as the [Search For Text Dialog](#).

A picture of the Model Explorer as it appears when showing the generator records is shown below. Besides showing case information display,


Explore Pane **Case Information Filterbar** **Recent / Tabs** **Case Information Toolbar**

The screenshot shows the 'Model Explorer: Generators' window. The 'Explore' pane on the left lists various components like 'Recent', 'Network', 'Generators', 'Impedance Correction', 'Line Shunts', 'Loads', 'Mismatches', 'Multi-Terminal DC', 'Switched Shunts', 'Three-Winding Transformer', 'Transformer Controls', 'Aggregations', 'Areas', 'Injection Groups', 'Interfaces', 'Islands', 'Multi-Section Lines', 'MW Transactions', 'Nomograms', 'Owners', 'Substations', 'Super Areas', 'Tie Lines between Areas', 'Tie Lines between Zones', 'Transfer Directions', 'Zones', 'Solution Details', 'Case Information and Auxil', 'Contingency Analysis', 'Fault Analysis', 'Optimal Power Flow', and 'Transient Stability'. The 'Case Information Filterbar' at the top shows 'Filter: Advanced' and 'Generator: NW-WA'. The 'Recent / Tabs' bar includes 'Gen Records (Filter: NW-WA)', 'Branches State', 'Interfaces', 'Zones', 'Areas', 'Switched Shunts', 'Loads', and 'Buses'. The 'Case Information Toolbar' on the right includes 'Records', 'Geo', 'Set', 'Columns', 'Options', and 'Find... Remove'. The main table displays generator data with columns: Number of Bus, Name of Bus, ID, Status, Gen MW, Gen Mvar, Set Volt, AGC, AVR, Min MW, Max MW, Min Mvar, Max Mvar, Cost Model, and Part. Factor. The table lists 32 generators, including CENTRALA, COWFALLS, MV-SVC, PORT ANG, ENSECH1, ENSECH2, ENSECH3, SUMAS L, TENASKA1, TENASKA2, TENASKAL, WHITHRN2, WHITHRN3, FREDONA1, FREDONA2, FREDONA3, FREDONA4, SAWMILL, LO BAKER, UP BAKER, CFALLGEN, KOMO K, MRPTGEN1, MRPTGEN2, MRPTGEN3, and MRPTGENL.

Number of Bus	Name of Bus	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor
1	40199 CENTRALA	1	Closed	10.00	0.00	1.02029	NO	NO	0.00	10.00	0.00	0.00	None	10.00
2	40307 COWFALLS	1	Closed	17.00	10.00	1.03913	NO	YES	0.00	35.00	-10.00	10.00	None	35.00
3	40307 COWFALLS	2	Open	0.00	0.00	1.03913	NO	YES	0.00	35.00	-10.00	10.00	None	35.00
4	40769 MV-SVC	1	Closed	0.00	49.00	1.08000	NO	YES	0.00	0.00	49.00	51.00	None	0.00
5	40841 PORT ANG	1	Open	0.00	0.00	1.00725	NO	NO	0.00	12.60	0.00	0.00	None	12.60
6	40841 PORT ANG	2	Open	0.00	0.00	1.00725	NO	NO	0.00	16.00	0.00	0.00	None	16.00
7	42011 ENSECH1	1	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
8	42012 ENSECH2	2	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
9	42013 ENSECH3	3	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
10	42014 ENSECHL	L	Open	0.00	0.00	1.01700	NO	YES	0.00	58.20	-12.51	30.16	None	58.20
11	42021 SUMAS 1	1	Open	0.00	0.00	1.00000	NO	YES	0.00	83.30	-41.34	61.55	None	83.30
12	42022 SUMAS L	L	Open	0.00	0.00	1.00000	NO	YES	0.00	48.70	-19.03	28.46	None	48.70
13	42031 TENASKA1	1	Open	0.00	0.00	1.02522	NO	YES	0.00	83.30	-22.36	55.19	None	83.30
14	42032 TENASKA2	2	Open	0.00	0.00	1.02522	NO	YES	0.00	83.30	-22.36	55.19	None	83.30
15	42033 TENASKAL	L	Open	0.00	0.00	1.02522	NO	YES	0.00	99.90	-38.76	55.32	None	99.90
16	42042 WHITHRN2	2	Closed	74.40	-11.28	0.98478	NO	YES	0.00	74.40	-19.92	47.74	None	74.40
17	42043 WHITHRN3	3	Closed	74.40	-11.41	0.98478	NO	YES	0.00	74.40	-19.92	47.74	None	74.40
18	42111 FREDONA1	1	Open	0.00	0.00	0.98478	NO	YES	0.00	104.00	-66.38	78.12	None	104.00
19	42112 FREDONA2	2	Open	0.00	0.00	0.98478	NO	YES	0.00	104.00	-66.38	78.12	None	104.00
20	42114 FREDONA3	3	Open	0.00	0.00	0.98478	NO	YES	0.00	50.00	-20.00	20.00	None	50.00
21	42115 FREDONA4	4	Open	0.00	0.00	0.98478	NO	YES	0.00	50.00	-20.00	20.00	None	50.00
22	42117 SAWMILL	1	Closed	25.00	7.00	1.01739	NO	YES	0.00	31.00	-7.00	7.00	None	31.00
23	42121 LO BAKER	1	Closed	59.60	1.00	1.02536	NO	NO	0.00	64.00	0.00	2.00	None	64.00
24	42124 UP BAKER	1	Closed	33.91	2.00	1.02681	NO	NO	0.00	47.00	0.00	2.00	None	47.00
25	42124 UP BAKER	2	Closed	33.91	2.00	1.02681	NO	NO	0.00	47.00	0.00	2.00	None	47.00
26	42126 CFALLGEN	5	Open	0.00	0.00	1.00000	NO	NO	0.00	15.00	0.00	0.00	None	15.00
27	42126 CFALLGEN	6	Closed	6.00	0.00	1.00000	NO	NO	0.00	15.00	0.00	0.00	None	15.00
28	42128 KOMO K	1	Closed	7.00	0.00	1.00638	NO	NO	0.00	14.00	-1.00	1.00	None	14.00
29	42131 MRPTGEN1	1	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
30	42132 MRPTGEN2	2	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
31	42133 MRPTGEN3	3	Open	0.00	0.00	1.02500	NO	YES	0.00	38.30	-8.00	20.00	None	38.30
32	42134 MRPTGENL	L	Open	0.00	0.00	1.02500	NO	YES	0.00	25.00	-6.00	12.00	None	25.00

Case Information Searchbar

Model Explorer: Fields Pane

The Fields Pane on the Model Explorer provides direct access to all the fields available for the case information display shown. The same features can also be accessed by choosing  **Display/Column Options** on the [case information toolbar](#) to open the dialog for [Configuring the Case Information Displays](#). A picture of the Fields Pane is shown at the bottom of this topic.

Dragging to Add Fields

After selecting fields from the Fields Pane, left-click and drag to add fields to the case information display. Fields will be added to the left of the column on which they are dropped. You will see red highlighting in the column headings indicating the position to which the fields will be added.

Dragging to Remove Fields

If you hold the CTRL key down and then left-click and drag a column heading from the case information display, you can then drop into the list of fields on the Fields Pane to remove the column.

Dragging to Move Fields

If you hold the CTRL key down and then left-click and drag a column heading on the case information display, you can move the columns around relative to one another. You will see red highlighting in the column headings to indicate the position to which the field is being moved.

Find Field

Click this button to search through the list of fields using a wild card search.

Add ->

Click this button and all selected fields in the Fields Pane will be added to the left of the presently selected columns in the case information display.

<-Remove

Click this button and all columns that have any cell presently selected will be removed from the case information display.

Expand and Collapse

Click this button to expand or collapse the folder view showing all the available fields.

Frozen Columns

The number of columns that are fixed and do not scroll when you scroll left or right in the display. Frozen columns will have a background color of gray instead of white. The number of frozen columns is 1 by default.

Reset to Factory Defaults

Click this button to reset all case information display properties to their default settings, including column and data field associations.

Fields Pane Location

By right-clicking on the Fields Pane or Explore Pane you can bring up the [Explore Pane Options](#). On this drop-down you can specify the Fields Pane Location to be either **Tab with Explore, Left Side**, or **Right Side**. The behavior for the various pane locations are shown as follows.

Tab with Explore

Fields

Find Field... Add ->

Available Fields <- Remove

- Frequency
- Long Name
- Name
- Name_Nominal kV
- Number
- Area
- Bus Shunt
- Calculated Field

Expand Collapse

Frozen Columns 1

Reset to Factory Defaults

Bus Records

Filter Advanced Bus Remove

	Number	Name	Area Name	Nom kV	PU Volt	Volt (kV)	Angle (Deg)	Load MW	Load
5	10007	ALLISON#	NEW MEXICO	115.00	1.03246	118.732	-44.02		
6	10008	ALLISON	NEW MEXICO	13.80	0.98365	13.574	-47.00	6.58	
7	10009	ALLISONT	NEW MEXICO	115.00	1.03453	118.971	-43.98		
8	10010	AMBROSIA	NEW MEXICO	115.00	1.04289	119.932	-48.47		
9	10011	AMBROSIA	NEW MEXICO	230.00	1.02619	236.023	-47.42		
10	10012	ANDERSON	NEW MEXICO	115.00	1.02963	118.408	-63.25		
11	10013	ANDERSON	NEW MEXICO	12.50	1.00065	12.508	-65.50	3.72	
12	10014	ARNO	NEW MEXICO	46.00	0.98799	45.447	-65.64		
13	10015	ARNO_1	NEW MEXICO	12.50	0.99179	12.397	-68.17	12.86	
14	10016	ARRIBA	NEW MEXICO	115.00	1.04694	120.398	-58.36		

Search Search Now Options

Right Side

Fields

Find Field... Add ->

Available Fields <- Remove

- Frequency
- Long Name
- Name
- Name_Nominal kV
- Number
- Area
- Bus Shunt
- Calculated Field
- Contingency Analysis
- Contingency Results

Expand Collapse

Frozen Columns 1

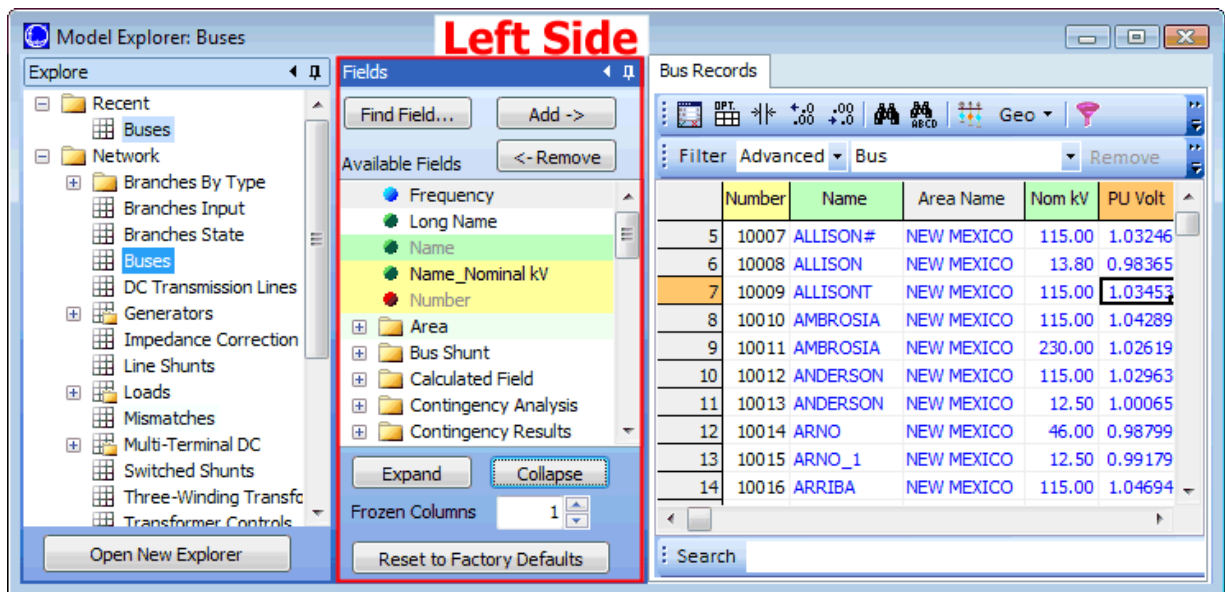
Reset to Factory Defaults

Bus Records

Filter Advanced Bus Remove

	Number	Name	Area Name	Nom kV	PU Volt	Volt (kV)	Angle (Deg)	Load MW	Load
5	10007	ALLISON#	NEW MEXICO	115.00	1.03246	118.732	-44.02		
6	10008	ALLISON	NEW MEXICO	13.80	0.98365	13.574	-47.00	6.58	
7	10009	ALLISONT	NEW MEXICO	115.00	1.03453	118.971	-43.98		
8	10010	AMBROSIA	NEW MEXICO	115.00	1.04289	119.932	-48.47		
9	10011	AMBROSIA	NEW MEXICO	230.00	1.02619	236.023	-47.42		
10	10012	ANDERSON	NEW MEXICO	115.00	1.02963	118.408	-63.25		
11	10013	ANDERSON	NEW MEXICO	12.50	1.00065	12.508	-65.50	3.72	
12	10014	ARNO	NEW MEXICO	46.00	0.98799	45.447	-65.64		
13	10015	ARNO_1	NEW MEXICO	12.50	0.99179	12.397	-68.17	12.86	
14	10016	ARRIBA	NEW MEXICO	115.00	1.04694	120.398	-58.36		

Search Search Now Options



Case Information Displays

Simulator offers numerous that provide a convenient, spreadsheet-like view of the power system and its components and are available regardless of whether the case has an associated oneline diagram or not. Many case information displays are available by using the [Model Explorer](#) button found on the [Case Information](#) ribbon tab, although these spread-sheet like views appear throughout the software as stand-alone displays and embedded inside other dialogs. Case information displays are available for buses, bus mismatches, generators, generator costs, ac lines, transformers, transformer impedance correction tables, dc lines, interfaces, areas, zones, schedules, and all other types of objects.

It is also possible to create [User-Defined Case Information Display](#) on the [Model Explorer](#).

Other screens, such as the [Area/Zone/Owner Filters](#) display, also fall into the category of case information displays. There are many different characteristics associated with the Case Information Displays, as discussed in the following topics:

[Colors and Cell Styles](#)

[Using Cell Handles](#)

[Case Information Toolbar](#)


[Case Information Filterbar](#)

[Sorting Records](#)

[Configuring the Case Information Displays](#)

[Finding Records](#)

Configuring the Case Information Displays

The content and format of all case information displays can be controlled through the Display / Column Options dialog. This dialog can be viewed by choosing  **Display/Column Options** on the [case information toolbar](#) or from the case information display's [local menu](#). The features on this display can also be accessed on the [Model Explorer: Fields Pane](#).

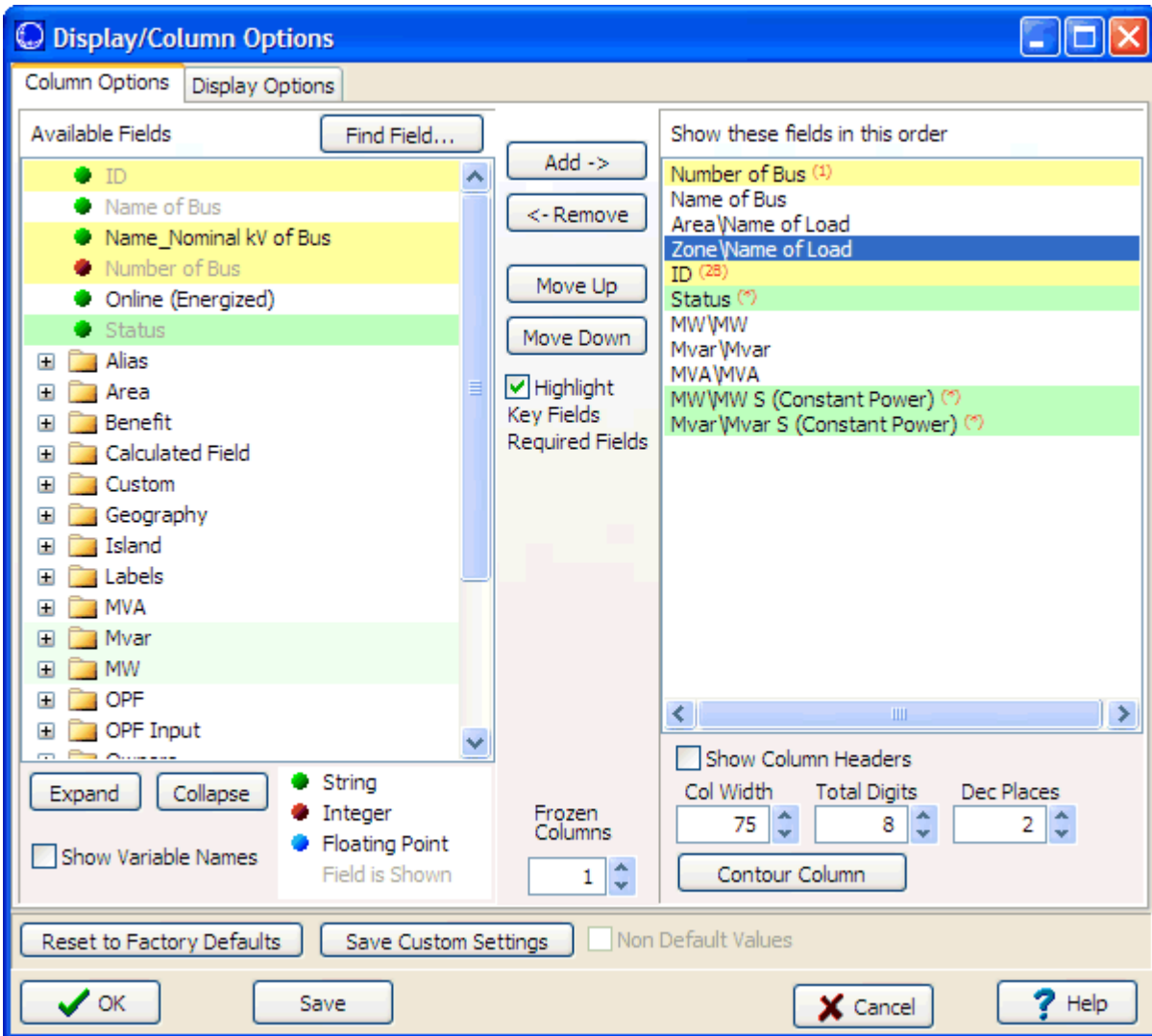
To reset all case information display properties to their default settings, including column and data field associations, click the **Reset All to Default Values** button.

To save your changes and close the dialog box, click **OK**. To save your changes without closing the dialog box, click **Save**. To undo the changes you have specified and close the dialog box, click **Cancel**. Finally, click **Help** to view the corresponding help screen.

While it is normally sufficient to customize the case information displays that comes with the program, it is also possible to create [User-Defined Case Information Displays](#) on the [Model Explore](#). This then provides the ability to switch between different sets of columns, filters, sorting etc... without requiring repeated customizations.

Column Options

The Column Options Tab is shown in the figure below.



Available Fields

This folder view contains all the fields available for addition to the respective case information display. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then add multiple fields to the Shown fields (see **Add->** below). Fields are generally organized into folder, so make use of the **Find Field** button if you have trouble finding the field you're looking for.

Note: when the mouse hovers over a field, a hint appears giving a longer description of the field.

Find Field

Click this button to search through the list of fields using wild card search.

Expand and Collapse

Click these button to expand or collapse the folder view showing all the available fields.

Show Variable Names

When this box is checked, the variable name corresponding to each column heading will be displayed in the list of available fields alongside the appropriate column heading.

Green, Red, and Blue dots

Also note that fields that the Green, Red, and Blue dots next to each string represents what type of value it is. Also, fields that are shown with gray text in the available field list represent fields that are already shown and in the list on the right.

Show these fields in this order

This list contains the fields which are presently shown on the respective case information display. Not that the field name is representing using backslash \ characters to signify the folders. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then remove multiple fields (see **Remove** below) or move the order of fields (see **Move** below).

Note: when the mouse hovers over a field, a hint appears giving a longer description of the field.

Column Width

Enter a new value to adjust the physical width of the column. This will affect all fields selected in the Show these fields list.

Total Digits

The number of digits to use when displaying values in the column (including the decimal point.) This will affect all fields selected in the Show these fields list.

Decimal Places

Number of total digits to the right of the decimal point. This will affect all fields selected in the Show these fields list.

Add > (Inserting Columns)

Click the **Add >** button to add all selected available fields to the end of the Show these fields list. You may also click on the selected Available fields and drag and drop the field on the Show these fields list. When drag/dropping the fields, all selected fields will be inserted just before the field that the mouse is over when you drop the fields. Once you have added fields you may then utilize the

< Remove (Removing Columns)

Click the **< Remove** button to remove the fields that are currently selected in the Show these fields list. You may also click on the selected fields in the Show these fields list and drag and drop the fields on the Available fields.

Move Up, Move Down (Moving Columns)

If you selected a block of fields in the Show these fields list, you may change the order of the list by clicking on the **Move Up** or **Move Down** button. You can also move the selection by left-clicking and dragging to move the columns around.

Also note that if you hold the CTRL key down and then left-click and drag a column heading on the case information display, you can move the columns around relative to one another. You will see red highlighting in the column headings to indicate the position to which the field is being moved.

Highlight Key Fields

Checking this option will highlight the *key fields* for the type of object displayed in the case information display presently being modified. The fields highlighted in Yellow in the two lists are either the Primary key fields (numbered) or the Secondary key fields (lettered). Note that some key fields, usually the Circuit ID, are both numbered and lettered, meaning they are used as both Primary or Secondary key fields. Either ALL numbered key fields OR all lettered key fields must be

included in the list of shown columns if you intend to export the data to either Excel or a text file, modify it, and paste it back into Simulator.

The fields highlighted in Green are the fields that would be necessary in Excel or an auxiliary file in order to create NEW objects of the type displayed when pasting from Excel or reading in an auxiliary file. If all of the green highlighted fields are not present in the external source, Simulator will not be able to create new objects that may be defined in that source, and will only paste information for objects that already exist in the present case.

Frozen Columns

The number of columns that are fixed and do not scroll when you scroll left or right in the display. Frozen columns will have a background color of gray instead of white. The number of frozen columns is 1 by default.

Display Options

Use Area / Zone Filters

Check this box to restrict the case information display's record set to cover only those areas and zones specified by the [area/zone/owner filters](#). If this box is unchecked, all records, regardless of zone or area, will be displayed. (The area/zone filters option obviously does not apply to the Area/Zone Filters information display.)

Advanced Filter

Clicking this button will open the [Advanced Filters Dialog](#), which will allow more detailed filtering on the display.

Zoom

You can zoom in and out on a case information display by using the keyboard short cut Ctrl+Up and Ctrl+Down. Alternatively you can zoom in out by setting a percentage on this dialog. This can also be modified under the Options Menu of the [Case Information Toolbar](#).

Use Custom Font / Row Height

If checked, then the Row Height and Custom Font specifications will be used. Changing either the Row Height or Custom Font properties will cause this box to become checked automatically. If you wish not to apply your custom specifications to the active case information display, uncheck this box; the default font and row height settings, as defined under the *Case Information Displays* tab of the [Simulator Options](#) dialog box or under the Options Menu of the [Case Information Toolbar](#), will be reapplied.

Row Height

Defines the height of the rows of the case information display.

Change Custom Font

Defines the font with which to display the records.

Set as Default

Sets the current Font and Row Height as the defaults.

Auto Size all Column Widths

Constrains all field widths to contain the widest data elements in each column.

Custom Field Descriptions

Custom Field Descriptions allow specification of custom floating point, integer, and string fields with user-defined column headers and field names. These fields can be added to [case information displays](#) for all power system case data and many supporting data types. Custom fields are currently not available for use with case information displays listing display objects. By default, there are two custom fields of each type available, but any number of each type of field can be defined for each type of data.

To view the Custom Field Descriptions, open **Custom Field Descriptions** found under the **Case Information and Auxiliary** folder on the [Model Explorer](#). The table lists the object types for which custom fields have been defined, how many of each type of field has been defined, and any custom field captions and headers for the fields. By default, there will be three entries; there are two fields each for floating point, integer, and string fields for all data types. These defaults will be used unless specific custom fields are defined for a particular data type.

To insert a new description, select **Records > Insert** from the case information toolbar. To modify an existing description, choose **Show Dialog** from the case information toolbar. Performing either of these actions will open the [Custom Field Description dialog](#). To remove an existing description, choose **Records > Delete**. *Default* descriptions cannot be removed.

The following fields are used for defining the custom field descriptions:

Object Type

Type of object for which the field is available. The *Default* type will be used for any data type that does not have its own definition. This field can only be set using the [Custom Field Description dialog](#) and cannot be changed once a description has been created.

Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set on the [Custom Field Description dialog](#) when inserting a new custom field description.

Number of Type

This value determines how many fields of the particular Field Type will be available.

Captions for Field (comma-separated)

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Captions for Headers are defined, the header captions will be used instead of the defaults.

Captions for Header (comma-separated)

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

Custom Field Description Dialog

The Custom Field Description Dialog is used to create and modify custom field descriptions for a selected object type. This dialog can be accessed from the [Custom Field Descriptions display](#) when inserting a new description by selecting **Records > Insert** or modifying an existing description by selecting **Show Dialog**.

	Field Captions	Header Captions
Floating Point 1	My Caption	My Header
Floating Point 2	Another Caption	Another Header
Floating Point 3	Extra Caption	Extra Header

Note: Blank entries for Field and Header captions mean that the default values will be used.

Object Name

Type of object for which the field is available. Select the type from the drop-down list or use the Find button to search through a list of available types. This field can only be set when creating a new description and cannot be changed once a description has been created.

Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set when inserting a new custom field description and cannot be changed once a description has been created.

Number of Type

This value determines how many fields of the particular Field Type will be available.

Clear Captions

Click this button to clear out any previously defined captions.

Field Captions

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Header Captions are defined, the header captions will be used instead of the defaults.

Header Captions

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

OK, Cancel

Click OK to accept the changes and close the dialog or Cancel to ignore the changes and close the dialog.

Calculated Fields

Calculated fields provide a means of performing an arithmetic operation (minimum, maximum, sum, average, median, variance, standard deviation, count) on a group of objects based on a selected field. Based on the object type of the group, a calculated field can be added to a [case information display](#) for a given object. The calculated field value that is displayed is then based on the calculated field parameters and how the object type in the calculated field relates to the given object.

For example, suppose that you want to find the largest line flow for all lines connected to a bus. A calculated field can be set up using the branch object type with a selected field of MVA (maximum) and the maximum operation can be performed on this field. This calculated field can then be associated with buses by adding this field to the [bus case information display](#). The resulting values for this field will show the maximum MVA flow for all lines connected to a given bus. When determining the value, only the branches that have a terminal bus the same as the given bus will be considered, and the maximum MVA value for all of those branches will be reported.

Optionally, filters can be associated with calculated fields to further specify what objects should be considered in the calculation.

The calculated fields case information display can be accessed from the [Model Explorer](#) under **Case Information and Auxiliary > Calculated Fields**. Existing calculated fields can be modified by selecting **Show Dialog** from the [case information toolbar](#), and new calculated fields can be added by selecting **Records > Insert** from the case information toolbar. Both actions will open the Calculated Field dialog shown below.

Calculated Field Dialog

Calculated Field

Calculated Field Name: Maximum MVA

Save Save As Rename Delete

Object Type: Branch

Branch calculations can be used with the following objects:

- 3W Transformer
- Area
- Bus
- Contingency
- Contingency Element Block
- DC Transmission Line
- Generator
- Global Contingency Actions
- Injection Group

Field: Find... MVA/MVA (maximum)

Use Absolute Value

Operation:

- Minimum
- Maximum
- Sum
- Average
- Median
- Variance
- Standard Deviation
- Count

Treat Blank Entries:

- As zeros
- Ignore

Objects to be included in the operation:

- All
- Only objects that meet the filter below

Set Filter Same As...

Select Filter Type: Branch

Pre-filter using Area/Zone/Owner Filters:

- Enabled (normally checked)

Logical Comparison:

- AND
- OR
- Not AND
- Not OR

Condition 1:

Find... Choose a Field between and ABS

Use Another Filter

Add >> Delete ...

Enable Field to Field Comparisons

OK Help Cancel

Calculated Field Name

This is the name of the calculated field as it will appear when added to a case information display. Use the **Save**, **Save As**, **Rename**, and **Delete** buttons to create new calculated fields without exiting the dialog. Switch between existing calculated fields by choosing a name from the drop-down box.

Object Type

Type of object that will be included in the calculation. The type of object determines which objects this field can be used with. This information is displayed to the right of the Object Type option.

Field

Select the field to use in the calculation by selecting a field name from the drop-down box or by clicking the Find button to use [advanced search](#) options to locate the field. Check the **Use Absolute Value** box to use the absolute value of the field in the operation.

Operation

Select the type of operation to perform on the object fields.

Treat Blank Entries

Depending on the field selected, some objects may not have entries. Select to either treat these blank entries **As zeros** or to **Ignore** them in the calculation.

Objects to be included in the operation

This option determines if **All** objects of the selected type will be included in the calculation or if they will be filtered by **Only objects that meet the filter below**.

If choosing to apply a filter, the filter can be defined by using the same options available when defining an [advanced filter](#). The **Set Filter Same As** button can be used to select an existing [advanced filter](#) whose actions should be applied to this filter.

Keep in mind that objects included in the calculation will first be determined by the object type. If for example branch is selected, then all branches will be considered. Setting a filter will limit the branches to only those branches that meet the filter. If the calculated field is used with buses, then the branches considered for a given bus will be only those branches that have a terminal bus the same as the given bus. If the calculated field is used with generators, then the branches considered for a given generator will be only those branches that have a terminal bus the same as the terminal bus of the given generator.

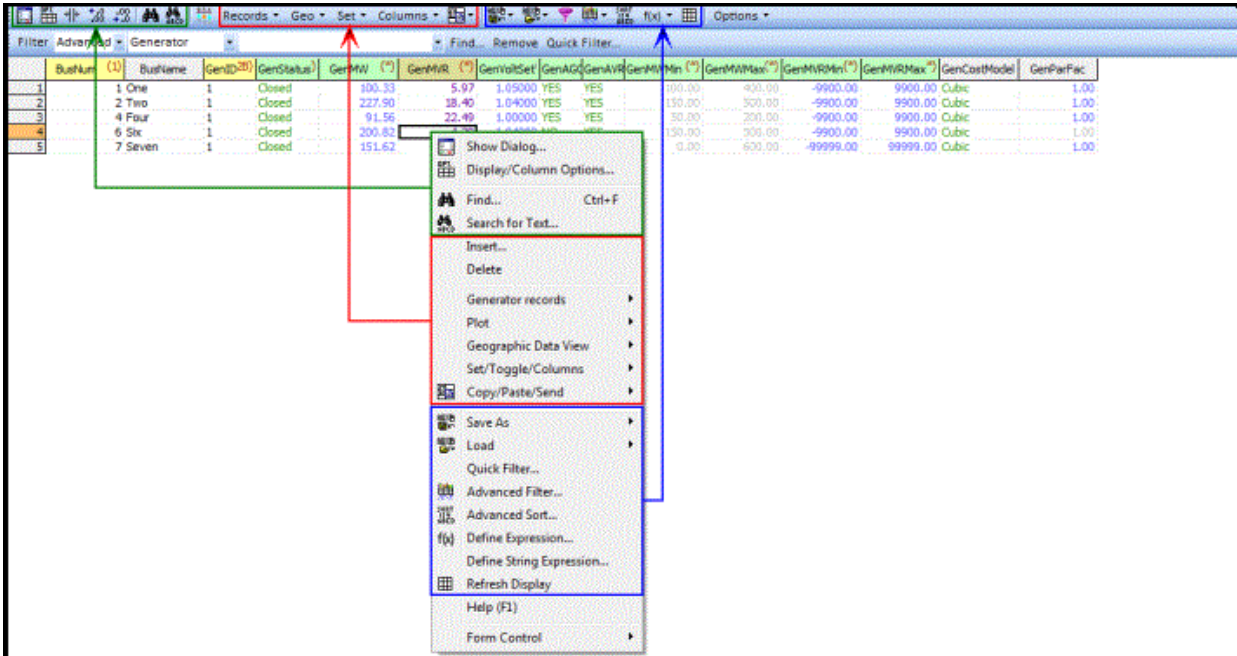
Case Information Customizations Display

To open the Case Information Customizations display, select **Case Information and Auxiliary > Case Info Customizations** from the [Model Explorer](#). If you have customized any of the case information displays for your case, you can see a list of the customized displays and their settings from this display.

Case Information Customization settings cannot be inserted manually or modified on this display. Those modifications must be done on the actual case information display. The primary use of this list display is to provide you a location to save and load case information customizations to an auxiliary file. Customizations can be loaded from an auxiliary file by right-clicking on the grid and choosing to load data from an auxiliary file. To save the customizations, right-click on this display and select **Save As > Auxiliary Data**.

Case Information Displays: Local Menu Options

All case information displays have a set of local menu options. The local menu can be brought up at any time by right clicking on case information display. Most of the actions and options available in the local menu are identical to the options available in the [Case Information Toolbar](#). This is depicted in the figure below.



For help on most of the actions and options available in the local menu, please see the [Case Information Toolbar](#) help. The other options that are also sometimes available on the right-click menu however are listed below.

Why is this field or line disabled?

This option will display a message showing the reason why the field is disabled. This option will be enabled only if the field is actually disabled.

Help

Display context-sensitive help for the case-information display.

Form Control

This menu option allows to control the case information display.

Print

You can print the contents of most of the case information displays by selecting Print from the local menu.

Close

Closes the case information display.

Make Top Left of Form Visible

Makes the top or the left of the case information display visible, if either the top or the left are not visible.

Shift Form Up

Moves the case information display up so the bottom of the case information display can be visible.

Maximize

Maximizes the size of the case information display.

Unmaximize (Restore)

Restores the size of the case information display.

Minimize

Minimizes the size of the case information display.

Case Information Displays: Colors and Cell Styles

The case information displays distinguish data field types by text color and cell style. Most of the entries on the case information display are colored using the following color convention. These colors can be customized using the Case Information Displays tab on the [Simulation Options Display](#), which can be invoked by selecting **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. **Simulator Options** is also available on the [Tools](#) and [Case Information](#) ribbon tabs.

There are three types of data fields:

Standard Fields Fields that cannot be modified directly from the case information display are colored black by default.

Enterable Fields Fields that can be modified are colored navy blue by default. When selected, a cell containing an enterable field will display a tiny filled square in its bottom right corner. This box is called a cell.

Toggleable Fields Fields whose values can be toggled are colored green by default. The values contained in toggleable fields are modified by double left-clicking on them or selecting the possible entries from the drop-down or dialog that becomes available when single left-clicking on the field. Like cells in enterable fields, cells in toggleable fields display a cell handle when selected.

Besides indicating field type, color coding is also used to highlight violations of branch flow, generator MW or MVR output, and bus voltage constraints. Fields that are either at a limit or violating a limit are colored red by default. This, too, is configurable from the Case Information Displays tab on [Simulation Options Display](#).

Case Information Displays: Using Cell Handles

When selected, cells corresponding to enterable and toggleable fields exhibit a small filled square in their bottom right corner called a cell handle. The cell handle may be used to propagate the value of the selected cell to other cells in the same field.

Suppose we have selected a toggleable or enterable cell and that we wish to copy its value to other records. Call this cell the *source cell* and its value the *source value*. To copy the source value to another record or records, perform the following steps:

- Drag the mouse onto the cell handle until the pointer becomes a crosshair.
- With the mouse pointer showing as a crosshair, click and hold the left mouse button.
- With the left mouse button depressed, drag the mouse up or down from the source cell to select a group of records to which to copy the source value. These destination cells will display a yellow background when selected in this manner.
- When you have finished selecting the destination cells, release the left mouse button.
- A message box will appear asking whether you want to change the values of the destination cells to the source value. Answer Yes to complete the copy.

Case Information Displays: Sorting Records

You can sort the entries on the case information displays by just about any field. To sort the records by a particular column, left-click on the column's heading. Left-click the column's heading again to reverse the sort order.

To sort the records by the absolute value of a field, hold down the shift key as you left-click on the field's heading.

Simulator also has a more advanced sorting tool, which can be accessed by selecting **Advanced Sort** from the [local menu](#). Advanced sort allows you to sort information based on values in more than one column of data. Advanced sort also allows you to sort based on the absolute value of numerical fields and by case sensitivity for string fields.

Case Information Displays: Finding Records

The Find Dialog is used to find the device of the specified type on the different Case Information displays. The Find Dialog is available on most case information displays from the local menu, which can be invoked by right-clicking in the grid.

Different Find dialogs exist for Areas, Buses, Interfaces, Lines and Zones. Note that the Bus Dialog is used to locate all bus objects, such as generators, loads, switched shunts and the bus itself. In general, the format of each dialog is similar, allowing you to find the desired object using either its number or name. The only exception is interfaces, for which no name is defined. The basics of the Find dialog are explained in the [Find Dialog Basics](#) help topic.

Saving Case Information Display Contents as HTML Tables

Simulator's [Case Information Displays](#) allow you to save their contents as HTML tables for display on the world-wide web. To do this, right-click on any row of the table and select **Save As HTML** from the resulting local menu. This brings up the [Table Format Dialog](#). Set the various table formatting options and click **OK**. Then, select the name of the file to which to save the HTML code. Finally, if a region of the table was selected, you will be asked if you want to save just the selected region as HTML. Indicate **Yes** to convert just the selected portion of the table, or click **No** to write the entire table as HTML.

HTML Table Format Dialog

The Table Format Dialog allows you to set various formatting options for HTML tables. This form is invoked when you try to save a [Case Information Displays](#) as HTML.

The Table Format Dialog is divided into two tabs, **Table Properties** and **Table Elements**.

Table Properties

This tab allows you to specify values for options that govern the appearance of the table as a whole. Here you can specify the following properties:

Border Weight

The thickness of the border to draw around each cell. Specify 0 to suppress the drawing of a cell border.

Horizontal Cell Spacing

The spacing to employ between cells that neighbor each other horizontally.

Vertical Cell Spacing

The spacing to employ between cells that neighbor each other vertically.

Table Width

The width of the table. If the Percent checkbox is checked, the width specifies the horizontal dimension of the table relative to the object that contains it on the screen.

Caption

The table title that will be printed directly above it on the web page.

Table Elements

These options control how the data will be centered in each cell. Choices include:

Horizontal Alignment

Controls how the text in each cell should be positioned horizontally. The default value is Left, but you may also choose to center or right-justify the data in each cell.

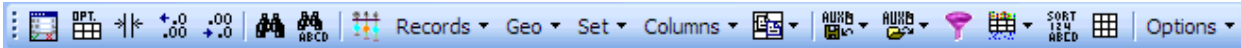
Vertical Alignment

Controls how the text in each cell should be positioned vertically. The default value is Middle, but you may also choose to align the text with the top or bottom edges of the cell.

If any of the numeric entries (Border Weight, Horizontal Cell Spacing, Vertical Cell Spacing, Table Width) are left blank or zero, your browser will employ its default settings for these values in rendering the table on the screen.

Case Information Toolbar

Directly above most case information displays, the Case Information Toolbar provides easy access to many [Case Information Display](#) options for formatting and customizing a list display. Note that all of the options are also available from the [right-click local menu](#) on the case information display as well.



Show Dialog

Selecting the Show Dialog option will invoke a dialog box containing more detailed information and settings regarding the corresponding system object. For example, clicking Show Dialog while a Bus Case Information Display is the active window opens the [Bus Information Dialog](#).

Display/Column Options

The contents and format of the information display can be controlled using the Case Information Display Dialog. See [Configuring the Case Information Displays](#) for more details.

Auto Size all Column Widths

Constrains all field widths to contain the widest data elements in each column.

Increase/Decrease Decimals

Adjusts the number of displayed decimal places for all cells in the selected column in a case information display.

Find

Use the Find local menu option to retrieve a record pertaining to a particular element. Choosing Find from toolbar opens the [Find Dialog Box](#), which is used to find records pertaining to an element identified by either number or name.

Search

Invokes the [Search for Text](#) dialog; allows you to search for specific text in a case information display.

Bus View Oneline

On many case information displays click this button to open the [Bus View Oneline](#) to the bus related to the presently active cell of the case information display.

Records Menu

The Records Menu contains actions that apply to the specific kinds of records displayed in the active case information display. [Custom Expressions](#), [Bus View Oneline](#), [Substation View Oneline](#), [Quick Power Flow List](#) and are always shown in this drop-down. For more details see the [Case Information Toolbar: Records Menu](#) Help.

Geo Data View Menu

The Geo Menu is only available on case information displays that show records related to the network model. For more details see the [Case Information Toolbar: Geo Data Menu Help](#).

Set and Columns Menu

Set contains actions for setting data or toggling data.

Columns contains actions that act on the column such as [Get Column Metrics](#). For more details see the [Case Information Toolbar: Set, Toggle, and Column Menu Help](#).



Copy, Paste and Send Menu

The Copy, Paste and Send Menu contains actions that copy and paste data through the Windows Clipboard. In addition it contains actions that send data directly to spreadsheet software such as Microsoft Excel or Open Office Calc. For more details see the [Case Information Toolbar: Copy, Paste, and Send Menu Help](#).



Save Auxiliary Menu

The Save Auxiliary Drop-Down contains actions that save data in the case information display to various formats. The [Auxiliary File Format](#) and Comma-Separate-Variable (CSV) formats are always available, but some records show other more specific formats. For more details see the [Case Information Toolbar Save Auxiliary Files Menu Help](#).



Load Auxiliary Menu

The Load Auxiliary Drop-Down contains actions that load data records related to the active case information display. The [Auxiliary File Format](#) is always a choice, but some records show other more record-specific formats. For more details see the [Case Information Toolbar: Load Auxiliary Files Menu Help](#).



Area/Zone/Owner Filters

Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.



Filtering Menu

The Filtering Drop contains actions related to filtering the records shown in the active case information display. For more details see the [Case Information Toolbar: Filtering Menu Help](#).



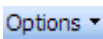
Advanced Sort

Allows the user to custom sort the information in the display based on desired criteria. See [Case Information Display: Sorting Records](#) for more information.



Refresh Display

Select this option to update the currently displayed data to match the present state of the system.



Options Menu

Invokes a drop down menu similar to the Case Information Display options dialog. See [Case Information Display Options](#) for more details.

Highlight Key and Required Field Column Headings

If this option is checked, the key fields needed for identifying objects will be highlighted in the case information display. These fields are important if you are planning on pasting data back into Simulator from Excel or reading data in from an auxiliary file. If the necessary key fields are not present in the Excel paste of auxiliary file, the data in those formats cannot be processed by Simulator.

Similarly, the Required Field Column Headings can also be highlighted in the case information display. In order for Simulator to create new objects by reading them from an auxiliary file or and Excel paste, the required fields must be present in the pasted data or the object cannot be created.

Headings

This option allows you to choose whether the column headings of the case information displays are the normal column headings or are the variable names of the type of data stored in each column. Variable names are the strings used when writing data out to an [Auxiliary File](#). Change the headings to variable names can be helpful when looking at or creating auxiliary files.

Show Grid Lines

Grid lines are shown by default in case information displays. Unchecking this box will remove the grid lines from the displays.

Show Header Hints

This option allows the hints for the current field to be displayed when moving the cursor over the column heading.

Word Wrap Headings

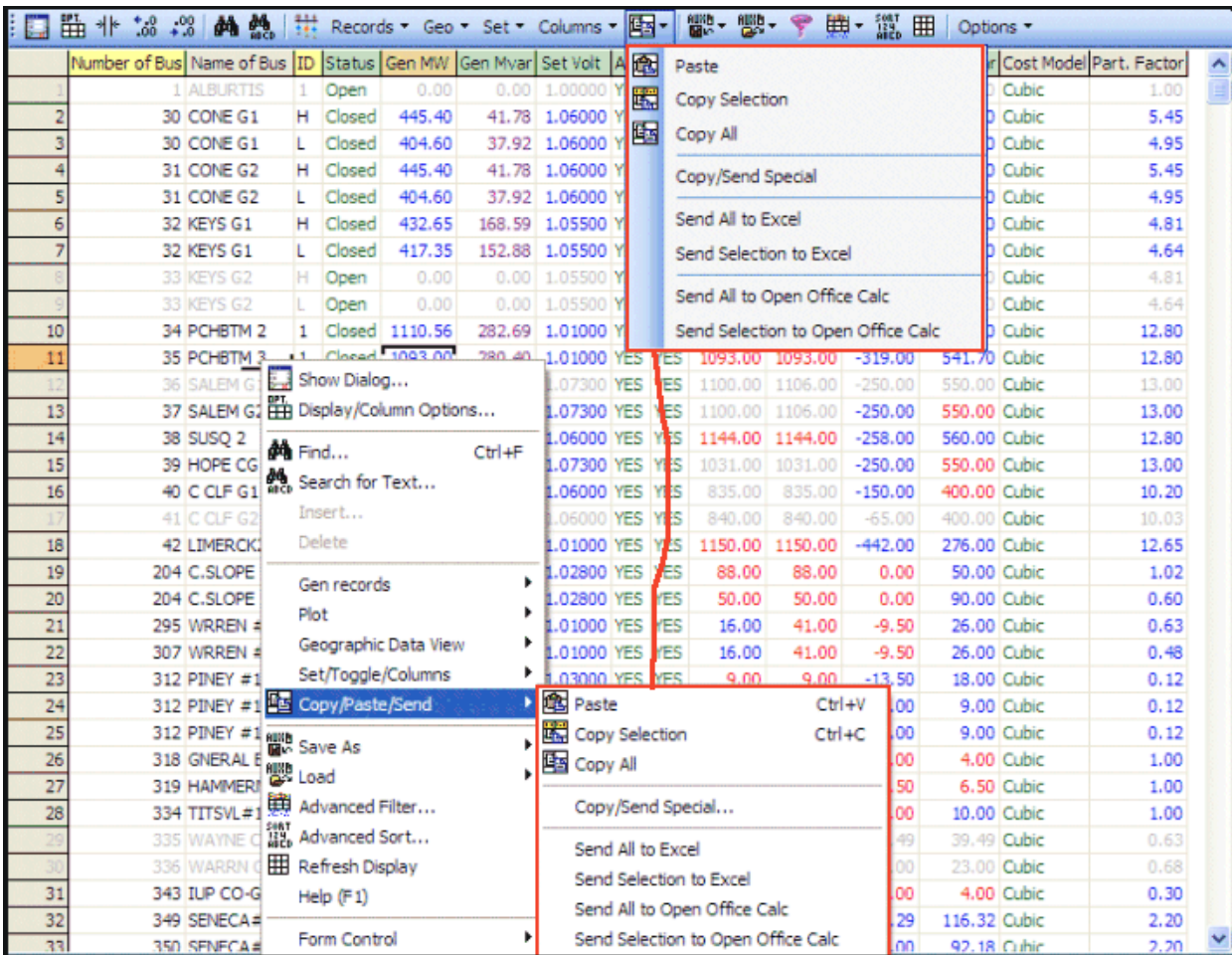
Check this box to wrap the heading column text, breaking it in several rows to fit the current column width. If unchecked, the column heading text will be displayed in only one row.

Highlight Objects if Selected? Field = YES

Check this option to automatically highlight all case information rows that represent a model object that has its "Selected" field set to YES. The row will then be highlighted using the color specified on this dialog. This option will apply to all case information displays representing all types of objects.

Case Information Toolbar: Copy, Paste and Send Menu

The [Case Information Toolbar](#) Copy, Paste and Send Menu displays the identical options as shown in the Copy/Paste/Send menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Paste

Select Paste from the local menu to copy a record set from the Windows clipboard into the case information display. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy Selection

The Copy Selection menu option copies the records selected in the case information display to the Windows clipboard, from which the selection can be copied into other programs such as Microsoft Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy All

The Copy All menu option copies the entire record set contained in the case information display to the Windows clipboard, from which it can be copied into other programs such as Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

Copy / Send Special

The Copy / Send Special menu option will open the a dialog that allows the user to set a few custom options before completing the data copy. The user can choose to copy all or a selection, and to copy the data either to the Windows clipboard or send it directly to Excel. Also, you can choose to copy or send the transpose. In addition, the user can choose whether to use the normal column headings or variable names (see the list of fields in [Data Argument List for Auxiliary Files](#)) as column identifiers. Lastly, the user can specify whether or not to include a row containing the type of object the data represents (Object Name) and a row containing the column headers for each column of data. Note that for pasting the information back into Simulator, the Object Name and Column Heading rows must be contained with the data to be pasted. If you have changed the settings on this dialog and wish to make them the case default settings, click on the Make Default for all Copy Actions button.

Send All to Excel (or to Open Office Calc)

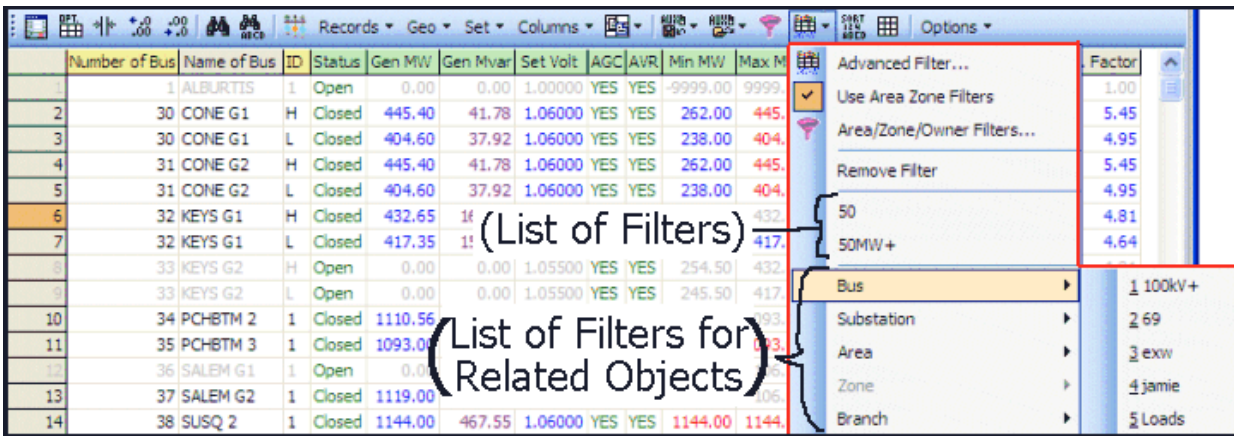
The Send All to Excel menu option copies the entire record set contained in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

Send Selection to Excel (or to Open Office Calc)

The Send Selection to Excel menu option copies the selected record set in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

Case Information Toolbar: Filtering Menu

The Filtering Menu, contained on the [Case Information Toolbar](#), gives access to several options regarding any [Advanced Filters](#) that can be applied to the active case information display. In addition it provides access to the [Area/Zone/Owner Filters](#). The drop-down menu is shown in the following figure.



The top of the menu has four primary buttons that appear on all case information displays. These four are as follows

Advanced Filter

Allows the user to custom filter the information in the display based on desired criteria. See [Advanced Filtering](#) for more information.

Use Area/Zone/Owner Filters

Filters displayed records by selections made on the [Area/Zone/Owner Filters](#) display.

Area/Zone/Owner Filters

Opens the [Area/Zone/Owner Filters](#) display for setting the areas, zones or owners for which the filter should apply.

Remove Filter

Click this button to remove the Advanced Filter from the present case information display. This does not delete the Advanced Filter definition, but only removes the reference to it from the active case information display. The filter can easily be reapplied after removing it. If you would like to permanently delete an Advanced Filter see [Advanced Filtering](#).

After the top section of the menu are two sections that may or may not be available for a particular case information display

(List of Filters)

This contains a list of advanced filters for the object type displayed in the active case information display. These filters would have already been created using either the [Advanced Filtering](#) dialog or by reading them from an [Auxiliary File](#). In the example image above the "50" and "50MW+" entries represent filters that apply to a generator object. If there are more than 10 advanced filters for the present object type, then a submenu will be shown for the object type as well. Also if an advanced filter is presently applied to the case information display, then the active advanced filter will be denoted with a check mark.

(List of Filters for Related Objects)

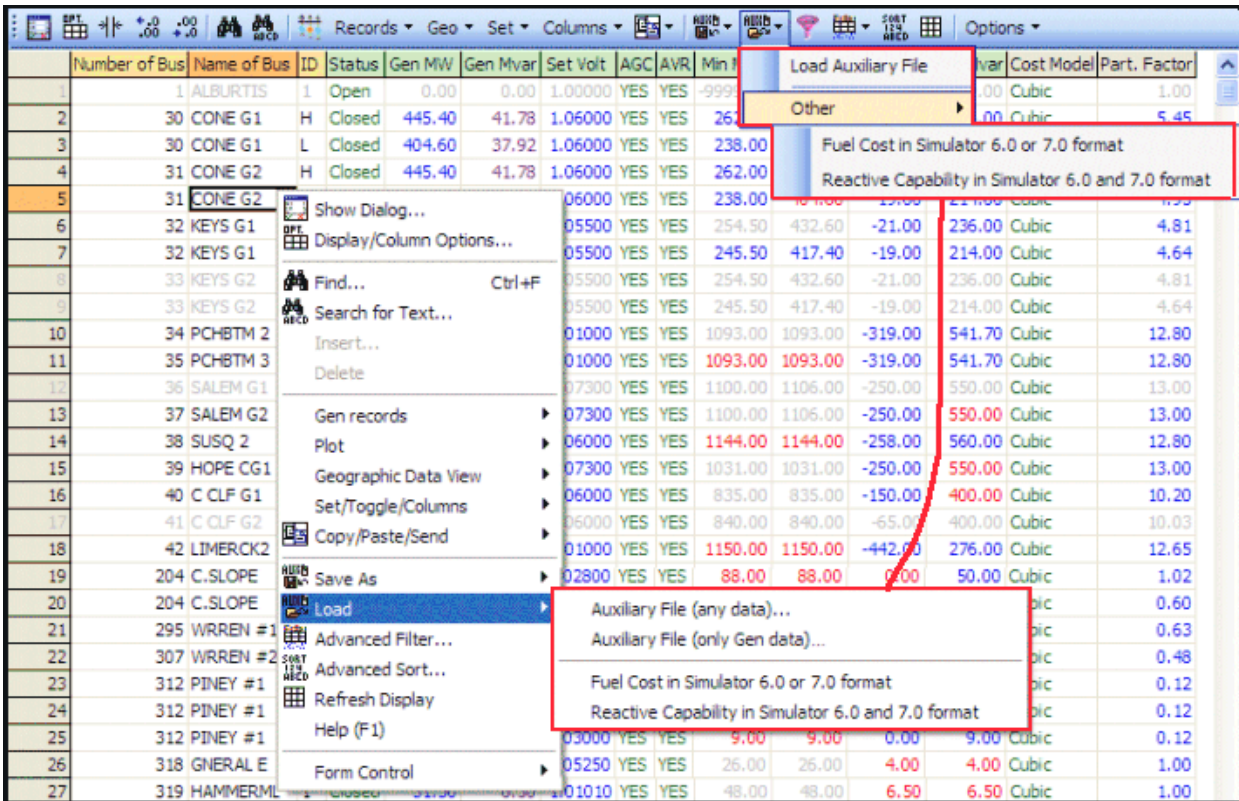
This contains a list of advanced filters whose object type is *related* to the type of record displayed in the active case information display. For example, the case information display shown in the figure above is showing generator records. Generator records allow you to filter the generator object using a Bus, Substation, Area, Zone, or Branch Filter. In the example above there are 5 filters defined for Bus objects (100kV+, 69, exw, jamie, and Loads) which can be used to filter the generator records. For more information on Related Object Filtering, see the [Advanced Filtering](#) dialog. Also if an advanced filter is presently applied to the case information display, then the active advanced filtered will be denoted with a check mark and the object type of that filter will be denoted with a grayed out check mark.

Case Information Toolbar: Geo Data View Menu

The Geo Menu, found on the [Case Information Toolbar](#), contains options related to opening the Geographic Data View Displays based on the objects selected in the case information display. For more information see the help on [Geographic Data Views](#).

Case Information Toolbar: Load Auxiliary Files Menu

The [Case Information Toolbar](#) Load Auxiliary Menu displays the identical options as shown in the Load menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will load the contents of an external file to the case information display.

Auxiliary File (Any Data)

This option allows loading an auxiliary file containing any data.

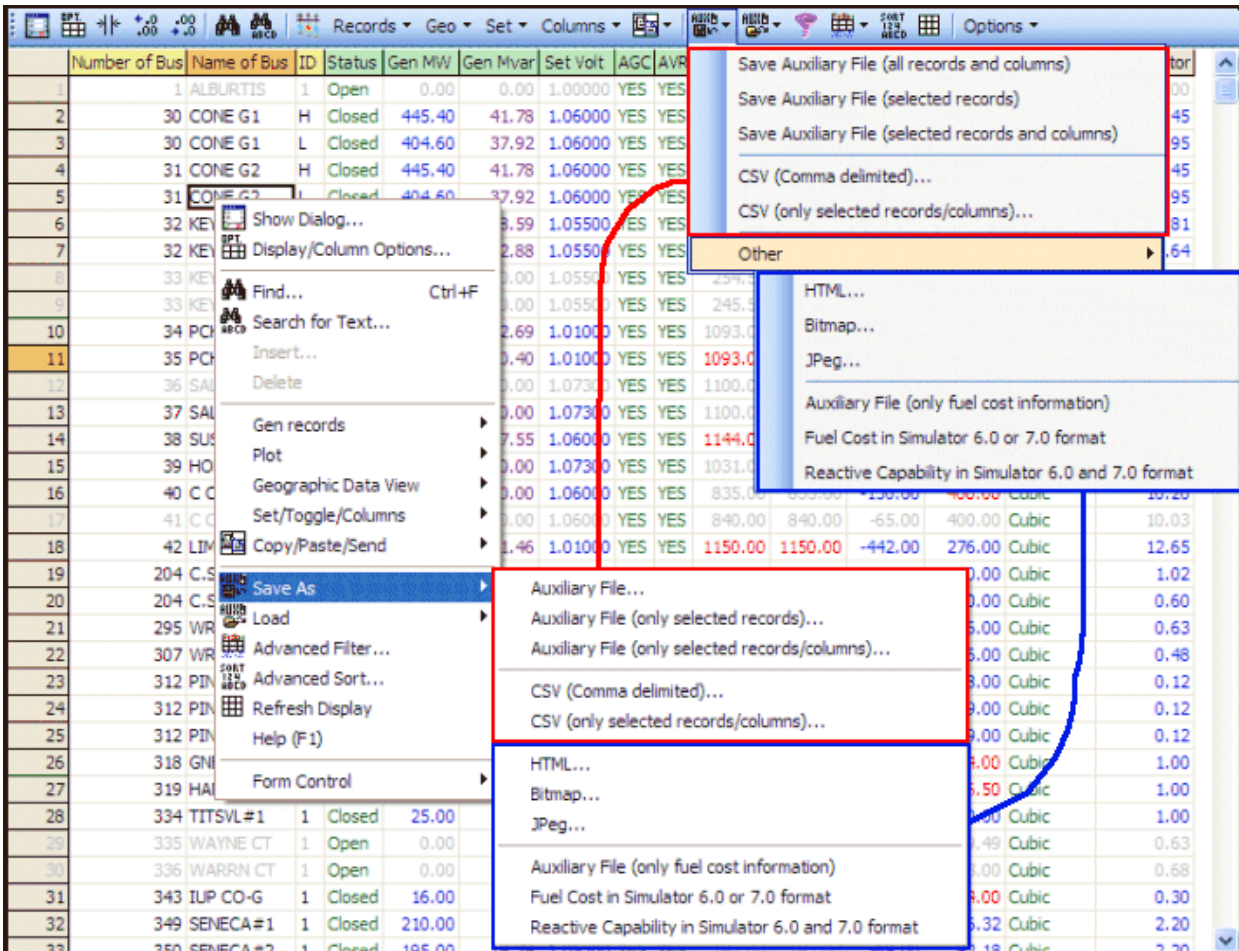
Auxiliary File (Only *Specific* Data)

This option allows loading an auxiliary file containing only data related to the current case information display.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see support for loading in older file formats previously supported.

Case Information Toolbar: Save Auxiliary Files Menu

The [Case Information Toolbar](#) Save Auxiliary Menu displays the identical options as shown in the Save As menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will save the contents of the case information display to an external file.

Auxiliary File

This option allows saving all the contents of the case information display to an auxiliary file.

Auxiliary File (only selected records)

This option allows saving the selected record set in the case information display to an auxiliary file.

Auxiliary File (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to an auxiliary file.

CSV (Comma Delimited)

This option allows saving all the contents of the case information display to a comma delimited file.

CSV (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to a comma delimited file.

HTML

You can save the entire table or selected records to an HTML file for viewing from an Internet browser. For more information, see [Saving Case Information Display Contents as HTML Tables](#).

Bitmap

This option allows saving the case information display into a Bitmap picture file.

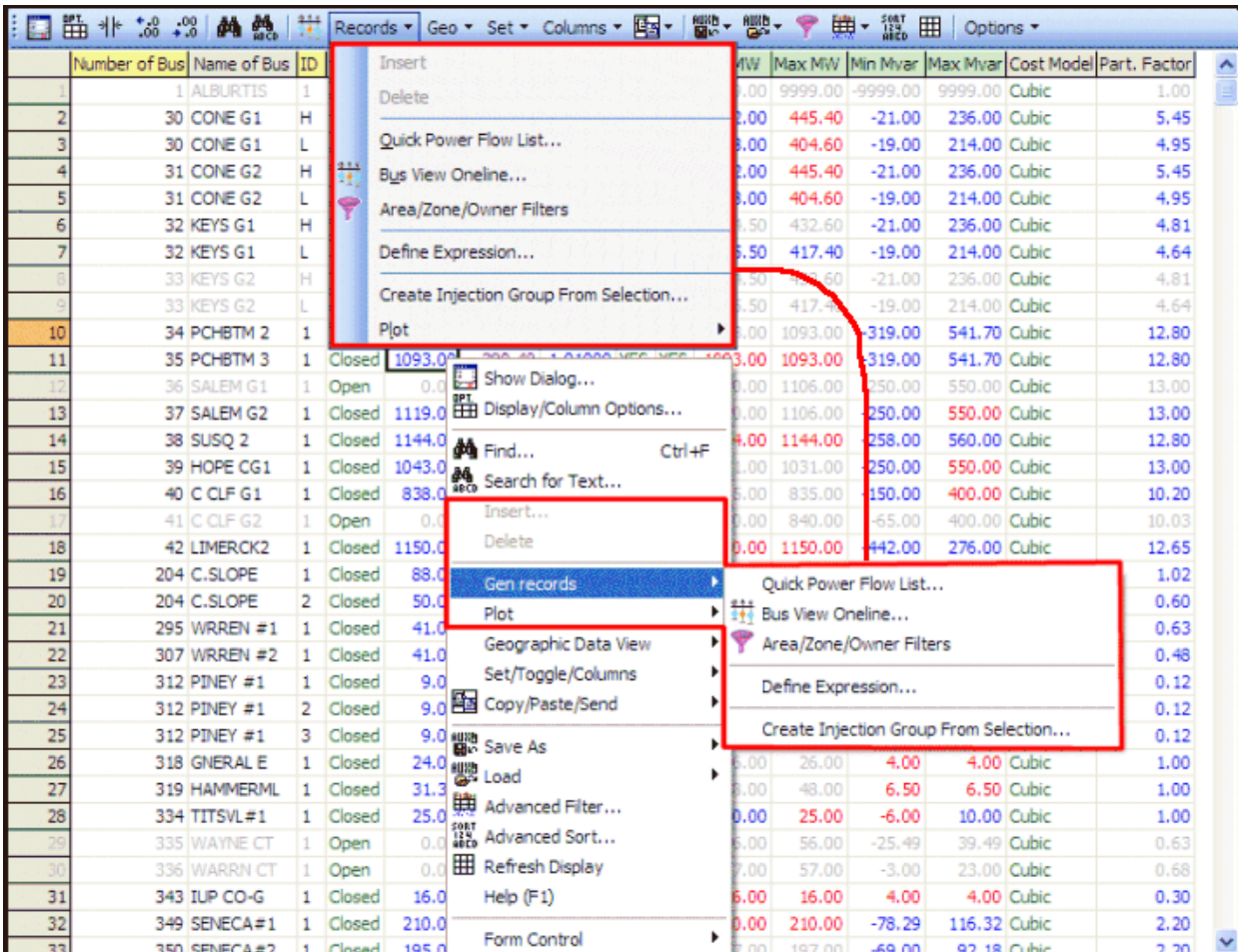
JPEG

This option allows saving the case information display into a JPEG picture file.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see a specific Auxiliary File format options which automatically saves all the data related to the cost information of the generator.

Case Information Toolbar: Records Menu

The [Case Information Toolbar](#) Records Menu displays a combination of the Insert/Delete buttons and the Records and Plot menus on the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Insert and Delete

Many records have an Insert and Delete related buttons. These are used for inserting new records and deleting records. Note that some records can only be inserted or deleted in Edit Mode. In such cases the Insert and Delete button will be grayed out when not in Edit Mode. Use the Bus View Online local menu option

Records Menu

Quick Power Flow List

Use the Quick Power Flow List local menu option to invoke [Simulator's Quick Power Flow List](#) tool.

Bus View Online

Use the Bus View Online local menu option to bring up the [Bus View Display](#), which illustrates how the selected bus is connected to the rest of the system.

Substation View Online

Use the Bus View Online local menu option to bring up the [Substation View Display](#), which illustrates how the selected substation is connected to the rest of the system.

Define Expression

Allows you to define [Custom Expressions](#) that are functions of other fields.

Pan to Object on Open Onelines

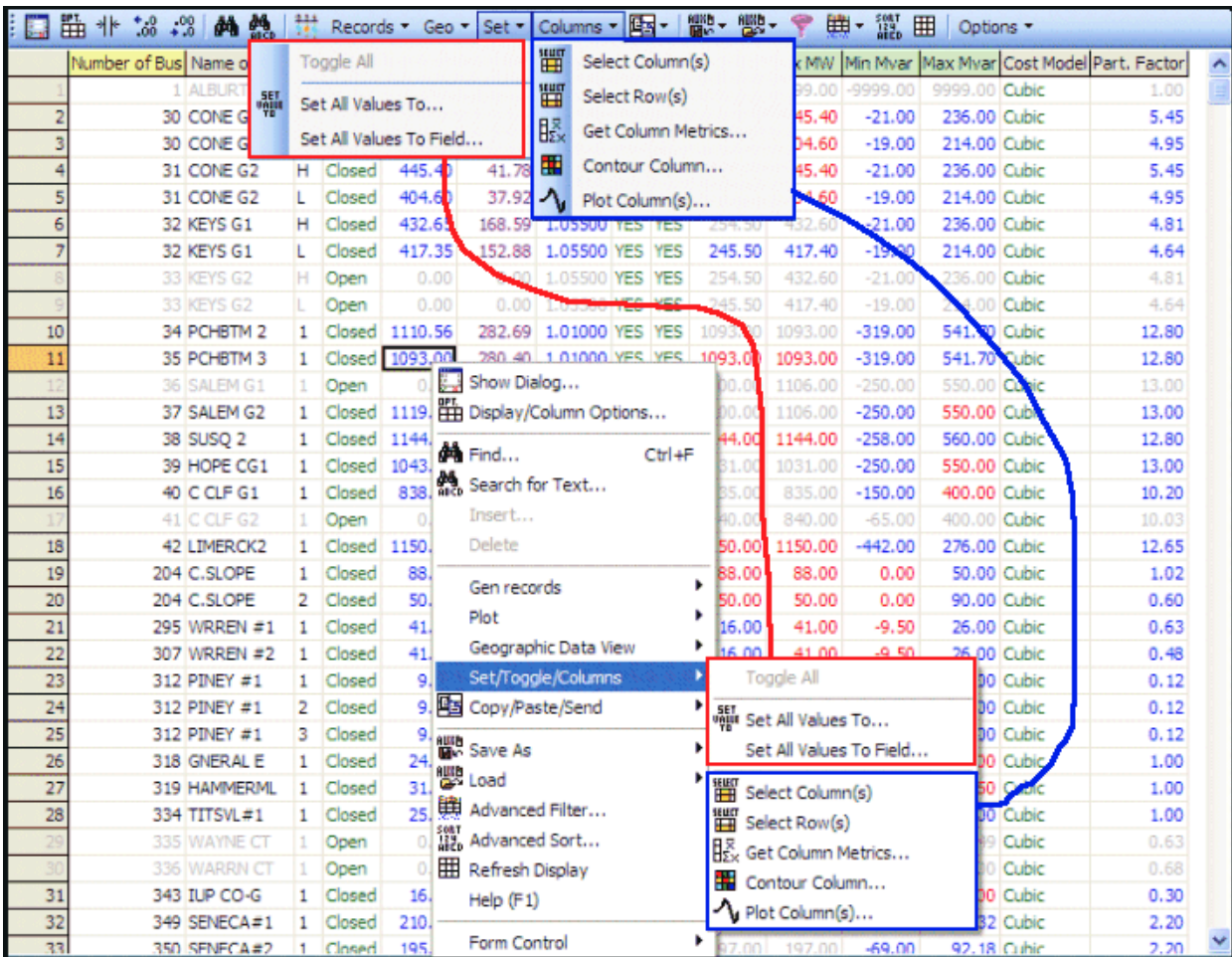
Selecting this option will move the view on any open online diagrams to focus on the device clicked in the table, if an object representing the device exists on the diagram.

Record-Specific Actions

At the end of the Record Drop-Down will be a list of options that are special for the particular kind of record shown in the active case information display. For example, in the image above for a generator case information display the option "Create Injection Group From Selection..." allows you to create a new injection group containing all of the presently selected generator records.

Case Information Toolbar: Set, Toggle, and Columns Menus

The [Case Information Toolbar](#) Set and Columns Menus display the identical options as shown in the Set/Toggle/Columns menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



Toggle

This option will be available for fields that are toggleable. The number and type of options available will change depending on the type of field being toggled. Some fields simply allow toggling between two YES/NO, TRUE/FALSE, or CLOSED/OPEN entries, while others allow a much larger list of choices. The toggle entries on the toolbar menus are used to toggle the field for all records.

Set All Values To

Click on this menu item to bring up a dialog which allows you to set all the values in a column to the same value. For instance you might change all the buses to have an angle of zero degrees.

A special trick that can be used in conjunction with this option is entering a string of the form "@variablename". When entering data into a field, if the string begins with the symbol @, then Simulator will parse the remaining part of the string to see if it matches one of the variable names associated with that type of object. Variable name strings are what are used in the Auxiliary File Format (see the list of fields in [Data Argument List for Auxiliary Files](#)). Essentially this trick allows

you to copy one column to another column. Using this trick, you could change the set-point voltage of all the generators in the case to be equal to the terminal bus voltage of the generator.

Set All Values To Field

Click on this menu item to bring up a dialog which allows you to select a field of this record type and then set all the values in a column to the this field. For instance you might change all the generator setpoint voltage to be equal to their present regulated bus voltage. This represents a user-interface method of employing the special trick using the "@variablename" string as discussed in the Set All Value To comments above.

Select Column(s)

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

Select Row(s)

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

Get Column Metrics

This option allows you to compute the metrics for the selected column. Choosing Get Column Metrics from the local menu will bring the [Grid Metrics Dialog](#). This option is only available for columns whose content is numeric.

Contour Column

Use the Contour Column local menu option to contour a column of the list display. Choosing Contour Column from the local menu will open the [Contour Column Dialog](#).

Plot Column(s)

Select this option to plot the values in any selected columns.

Case Information Filterbar

On the [Model Explorer](#), directly above the case information display is the Case Information Filterbar. Although the Filterbar is not shown by default on other case information displays not embedded in the Model Explorer, the Filterbar can be shown by right-clicking on the Case Information Toolbar and choosing **Filterbar>Top** or **Filterbar>Bottom**. The Filterbar is shown in the following image.



For some object types it is possible to filter based on data for a related object class instead of for the actual object class. For example, an Advanced Filter for a Bus may exist to only show buses with a Nominal Voltage larger than 138 kV. You may then want to use this filter on a case information display which shows generators. For object types which support this feature, the **Filter By Dropdown** and the **Filter Type Dropdown** will be available. For objects which do not support this, these dropdowns will not be visible.

Filter By Dropdown

This value will be either *Advanced* or *Device*. For more information about this setting see the [Advanced Filtering Dialog](#) help.

Filter Type Dropdown

This dropdown allows you to specify which class to base the filter on. For more information about this setting see the [Advanced Filtering Dialog](#) help.

List of Filters or Devices Dropdown

When the **Filter By Dropdown** is set to *Advanced*, this dropdown will contain a list of all the presently defined advanced filters for the object type specified in the **Filter Type Dropdown**.

When the **Filter By Dropdown** is set to *Device*, this dropdown will contain a list of all objects of the object type specified in the **Filter Type Dropdown**.

When clicking on this dropdown the appropriate list will be shown and by choosing an entry from this dropdown the respective advanced or device filter is applied to the case information display. Also, if the dropdown contains more than 1,000 entries in it, clicking on this dropdown will instead have the same effect as clicking on the **Find... Button**.

Find... Button

Click on this button to open the [Advanced Filtering Dialog](#) for the case information display.

Remove Button



Click this button to remove the [Advanced Filter](#) or [Quick Filter](#) from the present case information display. This does not delete an advanced filter definition, but only removes the reference to it from the active case information display. The filter can easily be reapplied after removing it. If you would like to permanently delete an Advanced Filter see [Advanced Filtering](#).

Quick Filter

Click this button to set a [Quick Filter](#) using the column in which the cursor is currently set in the display.

Area/Zone/Owner Filters

The Area/Zone/Owner Filters Dialog allows you to filter the information shown on the [case information displays](#) and other dialogs by area, zone or owner. There are a several ways to open the Area/Zone/Owner Filters Dialog.

- Go to the [Case Information](#) ribbon tab, and choose  **Area/Zone Filters** from the **Case Information** ribbon group
- Click on the button  on the [Case Information Toolbar](#)
- There are also many buttons on user interface dialogs throughout the program that invoke this dialog as well.

For small cases, you will usually not need to use this filtering capability, but it can be very useful for large cases. The filters display lists each area, zone and owner in the case, the number of buses in each, the range of bus numbers contained in each, and whether or not information about that area, zone or owner should be displayed. In order for a device to be displayed, its area, zone and owner Shown property must be set to yes. For devices which can belong to more than one area, zone, or owner (for instance a transmission line that connects two areas, i.e. a tie line), then the device will be displayed if any end of the device meets the area/zone/owner filter.

You can switch between displaying the filters for the case areas, zones or owners by clicking on the associated tab.

The area/zone/owners filters list is itself a case information display and therefore shares many of the same [local menu](#) options and characteristics. Using the local menu, you can search for an area, zone or owner by number or by name, copy records to and from other applications, and send the records to a printer. You can inspect an area, zone or owner by selecting *Show Dialog* from the local menu, which invokes the [Area Information](#) , [Zone Information](#) or [Owner Information](#) Dialogs. You can also change the format and content of the filters display by selecting the [Display Column/Options](#) item from the local menu. The records can be sorted by any of its component fields simply by clicking the corresponding column's heading.

The display's only enterable field is the one entitled *Shown*, which may assume only the values Yes and No. For example, if the Area/Zone/Owner Filters setting for an area is *No*, then any case information display configured to enforce area/zone/owner filters will omit the area's elements from the resulting record set. You can specify whether a particular case information display enforces filters using the display's [Display/Column Options Dialog](#).

Double-click on a cell in the *Shown* field to toggle its value. Use the [cell handle](#) to propagate a particular value to multiple areas, zones or owners, or use the *Toggle All Yes* or *Toggle All No* local menu options to set the values of all area, zone or owner records.

Advanced Filtering

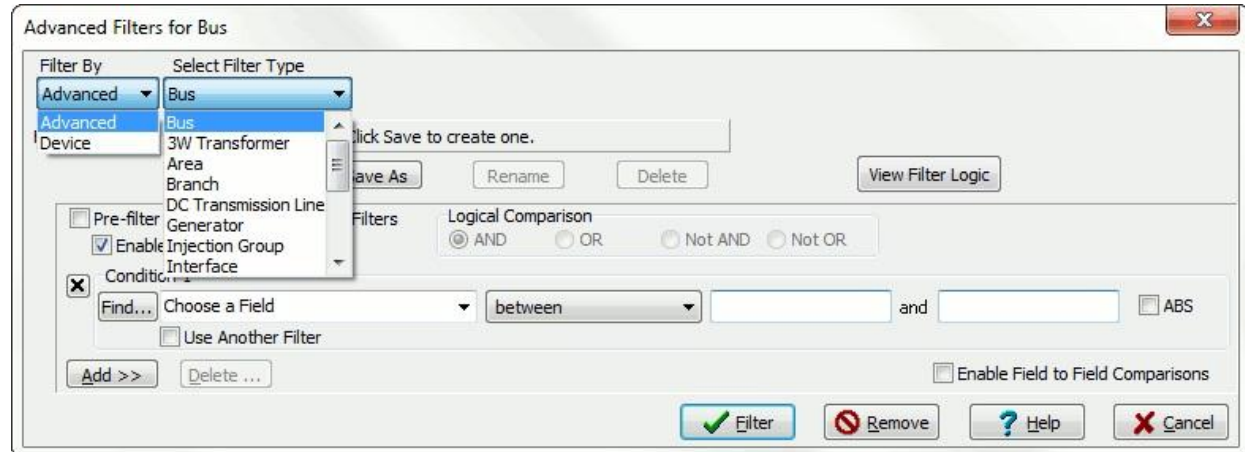
Filtering by areas, zones and owners using [Area/Zone/Owner Filtering](#) is a quick and simple way to filter, however Simulator also contains the ability to perform custom filtering on case information displays as well. Advanced Filtering is accessed by clicking on the [Case Information Toolbar Filtering Menu](#) and choosing Advanced Filter or right-clicking on a [Case Information Display](#) and choosing Advanced Filter from the local menu. This brings up the [Advanced Filters Dialog](#), which allows you to custom filter the information in the display. Advanced Filters require a filter name, and are stored with your power system case.

Note that you can also use the [Quick Filter](#) option on a case information display to set conditions for a filter just as you would for an advanced filter, without requiring a filter name. Quick filter conditions are not saved with the load flow case.

A [Case Information Display](#) will have the phrase "Filter: *Name of Filter*" in its caption if an Advanced Filter has been applied to it. To remove a filter from a [Case Information Display](#), choose Remove Filter from the Case Information Toolbar Filtering Menu or bring up the [Advanced Filters Dialog](#) and click on **Remove**. Note that Remove does NOT delete the filter, but just stops using it to filter the data. You can always remove the filter temporarily and then come back and reapply the filter. If you want to Delete a filter, you must bring it up in the [Advanced Filters Dialog](#) and click on **Delete**.

Advanced Filters Dialog

The first thing to choose on the Advanced Filter dialog is what to **Filter By** and the **Filter Type**.



Filter By

Filter By can be set to either **Advanced** or **Device**.

Advanced Filtering allows you to define a set of logical comparisons for the various fields which must be met. For example you may want to only see Buses that meet the conditions "Nominal Voltage > 100 kV" AND "MW Load > 100 MW". For more help on Advanced Filtering see the topic [Advanced Filtering: Advanced](#).

Device Filtering allows you to directly use one of the power system model objects as a filter. The relationship between the object type being listed and the object type of the device filter determines how filtering is done. For example, if you use an Injection Group as a device filter to filter a list of Generators, then you will get only the generators which are inside the Injection Group. As another example, if you are using an Injection Group as a device filter to filter a list of Branches, then you will get only branches that are connected to the terminal bus of any generator, load, or switched shunt contained in the injection group. Device filtering can be used instead of more complicated Advanced Filters. For more help on Device Filtering see the topic [Advanced Filter: Device](#).

Select Filter Type

When defining an Advanced Filter, normally this type is the same as the object type for which your are defining a filter. When defining a Device Filter, normally this type will be different.

By changing the drop-down next to Select Filter Type, it is possible to filter based on data for a related object instead of for the actual object. For example, an Advanced Filter for a Bus may exist to only show buses with a Nominal Voltage larger than 138 kV. You may then want to use this filter on a case information display which shows generators. This is possible by changing the Filter Type to be a Bus when showing the Filter Dialog on the Generator case information display. Just click the drop-down to see which related objects' filters can be used for that particular situation.

View Filter Logic

Select [View Filter Logic](#) to show a graphical display to visualize the logic diagram for an Advanced Filter.

Note Regarding Auxiliary Files: Throughout auxiliary script commands, the ability to specify an advanced filter by name is available. In this context, if the use of an Advanced Bus filter is desired in a place where an Advanced Generator filter is needed, then use the string "<BUS> filtername" to

represent the filter. If the use of a device filter is desired, then use the string "<DEVICE> objecttype 'key1' 'key2' 'key3'" to represent the device filter.

Advanced Filters View Filter Logic Graphical Display

The **Advanced Filter Logic Display** feature provides a graphical display which allows you to quickly browse information about an [Advanced Filter](#) and the corresponding conditions. The advanced filter logic displays enable convenient Advanced Filter-by-Advanced Filter navigation through the complete list of model filters. From the advanced filter logic display, you can find out an advanced filter name and Logical Comparison object, and conditions names that are part of the particular advanced filter. Moreover, you can find out all information associated with the advanced filter by pressing right click with the mouse and directly invoking their associated information dialogs. The advantage of the advanced filter logic displays, is that they are auto-created logic diagrams.

Below this top panel sits the actual advanced filter display. The advanced filter you have chosen to inspect, is represented as the last advanced filter to the right of the display. When you drag the mouse over one of the neighboring advanced filters symbols, it turns into a pointing finger, then by clicking the left mouse button when the mouse cursor is in this shape redefines the target advanced filter to be the advanced filter whose symbol you just clicked. The advanced filter view display is redrawn to show the same sort of display for the newly chosen target. Right-clicking on the advanced filter view display's background will generate the local menu.

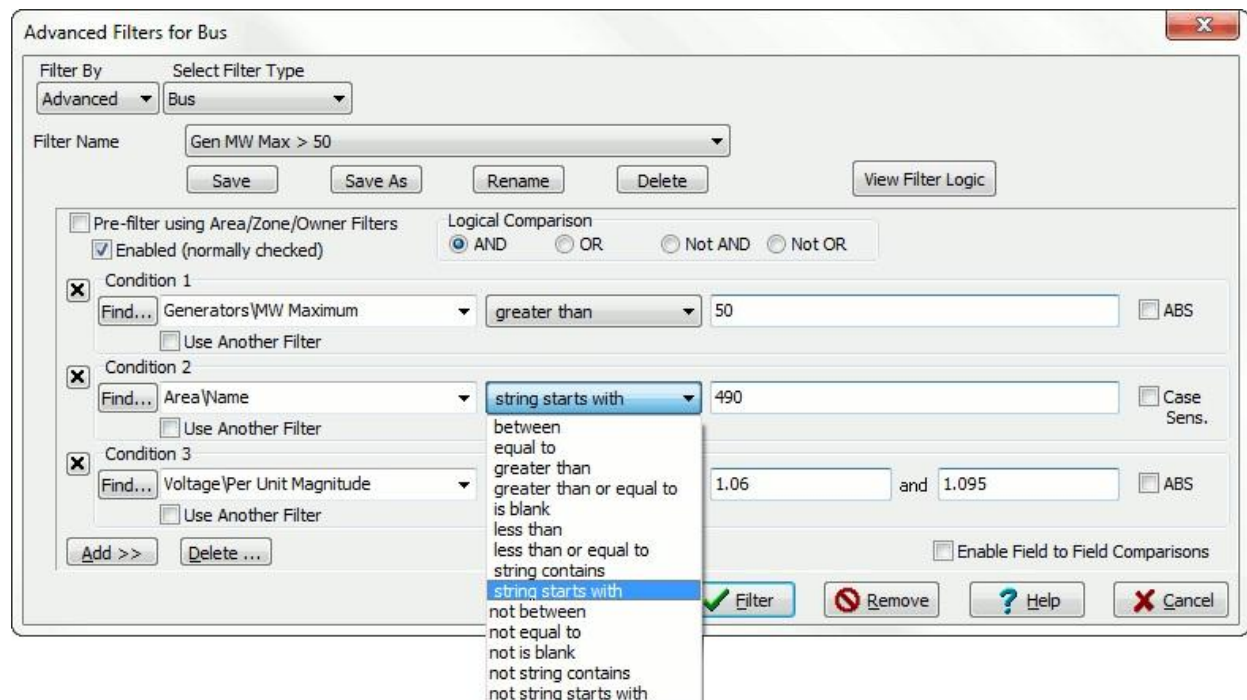
The advanced filter background color can be red or green. Green means that the criteria is satisfied and the evaluation of the model filter is true. Red means that the evaluation is false. Repeated colors of the Advanced Filter Names and Conditions represent that the particular Advanced Filter or Condition is used more than once in the Advanced Filter Display.

The advanced filter view display can be generated using the following method:

- Pressing the "**View Filter Logic**" button in the [Advanced Filter Dialog](#) Dialog.

Advanced Filtering: Advanced

The [Advanced Filters Dialog](#) **Filter** set to **Advanced** is shown in the following figure.



When you open this dialog, you will only see filters that have been defined for the type of object you are trying to filter (e.g. Bus, Generator, Interface, etc...). You can choose a filter from the Filter Name drop-down box showing the list of filters available, or you can create a new filter. To make a new filter, simply click on Save As to save a copy of the present filter under a new name and then specify the properties of the filter as discussed below. When you have specified the filter as you wish, click **Filter**.

Note that advanced filters are stored with the case file when the case is saved. In addition, the filters can be exported to a Simulator [Auxiliary File](#) for storage and import into other cases. A list of all advanced filters defined for a case can be viewed in the [Advanced Filters case information display](#).

To create an Advanced Filter you must specify the following things:

Filter Name

A string that describes your filter so that you can call it up from other forms in the future.

Condition 1, Condition2, etc...

Describes the conditions of your filter. To define a condition

- Specify the field you are filtering. By default, the fields in the drop-down list are limited to the fields presently shown as columns in the display. If you wish to choose from a list of all fields for the object, change the radio button setting to **Show All Fields**. It is very useful to use the **Find...** button to more easily search for the field you are interested in.
- Specify the comparison operation such as "**between**" or "**greater than**".
- Specify the values the field is compared to. Depending on the comparison operation, either one or two values are needed.
- Select **ABS** to use the absolute value of the field value in the comparison operation. For Fields that are strings, select **Case Sens.** to make the comparison case sensitive.

Note: The comparison operation "within integer range list" uses the same format as described in [Entering a Range of Numbers](#).

Note: To add or delete conditions click on the **Add>>** or **Delete...** button. You can also delete an individual condition by clicking on the **X** button next to the condition.

Note: There is no limit to the number of comparisons you use for the Advanced Filter.

Logical Comparison

The following describes how the Filter uses the Conditions which are specified

- AND means that all conditions should be true
- Not AND is the opposite of AND (i.e. any one of the conditions can be false)
- OR means that any one of the conditions can be true
- Not OR is the opposite of OR (i.e. all of the conditions must be false)

For advanced logical comparisons within the same filter, refer to the important note below.

Pre-Filter using Area/Zone Filters

When this box is checked, data is filtered first by the [Area/Zone/Owner Filters](#) and then by the Advanced Filter, therefore the data must meet both the Area/Zone/Owner Filters and the Advanced Filters in order to be shown. When this box is unchecked, the [Area/Zone/Owner Filters](#) are ignored.

Enable Field to Field Comparisons

When this box is checked, it is possible to compare two fields of the device. In order to do so, select **Field** and click on **Find** to select the second field to be compared. The following depicts what a particular condition looks like when the enable Field to Field comparisons checkbox is checked.

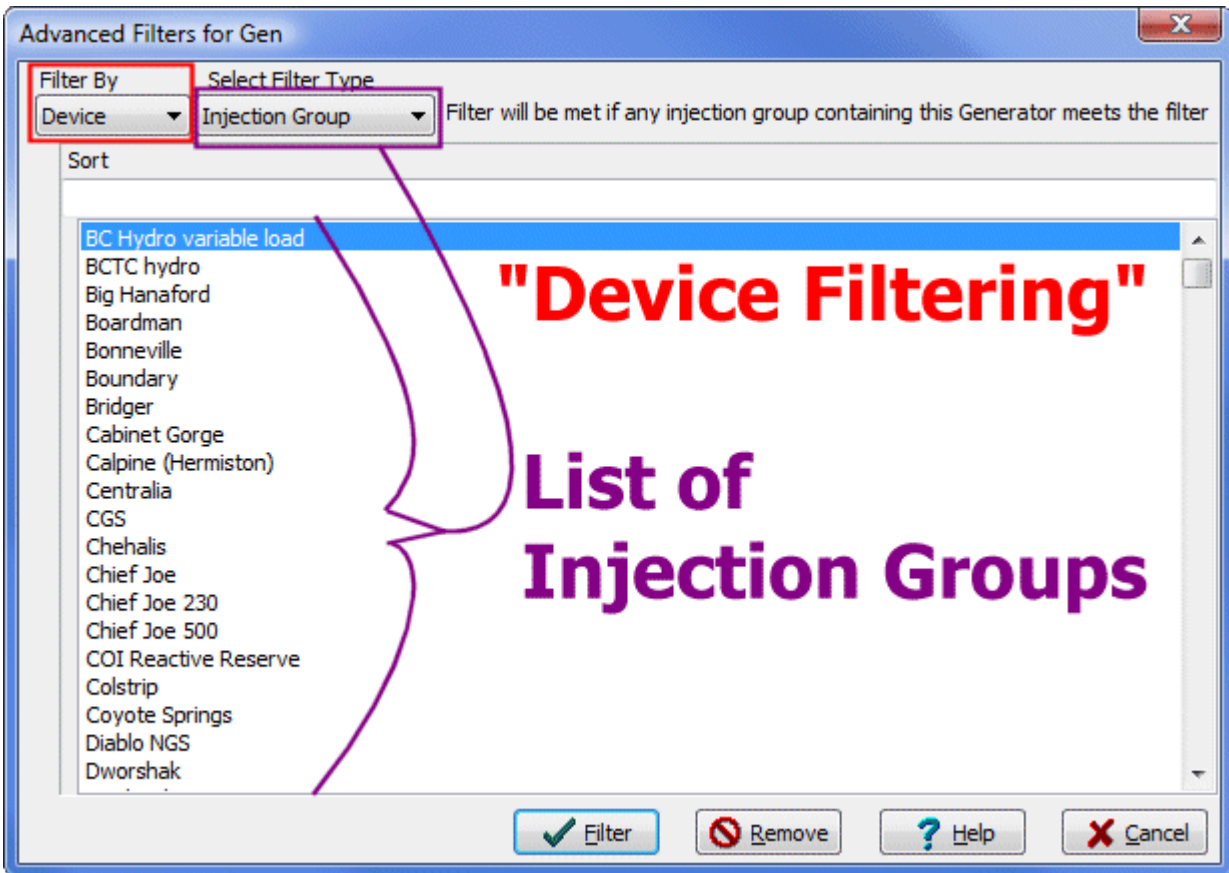
The screenshot shows a dialog box titled 'Condition 1' with a close button (X) in the top left. It contains a 'Find...' button, a dropdown menu with 'Generators\MW' selected, a comparison operator dropdown with 'greater than' selected, another dropdown menu with 'Generators\MW Maximum' selected, and an unchecked checkbox labeled 'ABS'. Below these is a 'Use Another Filter' checkbox. At the bottom left are 'Add >>' and 'Delete ...' buttons. On the right side, there is a 'Field' dropdown menu, a 'Find' button, and a list showing 'Constant' and 'Field' (which is highlighted). At the bottom right, there is a checked checkbox labeled 'Enable Field to Field Comparisons'.

Important Note: Once you choose a condition comparison type, that type is used for all conditions in the filter. Therefore if you wish to use the AND condition, all conditions you define will be applied using AND. There is a way to combine different conditions within the same filter. This is accomplished by allowing nested filters. In other words, one condition of a filter can be that another filter is met. To refer to one filter from within another, check the box **Use Another Filter**. Then choose whether the condition is to meet the filter or **Not** meet the filter, then choose the filter (which must have been previously defined) from the drop down list of the third box, or by clicking the Find button. This allows you to define some conditions using one logical comparison in one filter, and then use that filter to combine those conditions with other conditions using a different logical comparison.

For example, consider the logical comparison of **A and (B or C)**. To replicate this, you would define one advanced filter (AF1) that contains the logic **B or C**. Then you can define an advanced filter (AF2) that uses AF1 as **A and (AF1)**.

Advanced Filtering: Device

The [Advanced Filters Dialog](#) **Filter by** set to **Device** is shown in the following figure



Device Filtering allows you to directly use one of the power system model objects as a filter. The relationship between the object type being listed and the object type of the device filter determines how filtering is done. For example, the image below shows the use an Injection Group as a device filter to filter a list of Generators. The result of using this filter will be only the generators which are inside the Injection Group.

As another example, if you are using an Injection Group as a device filter to filter a list of Branches, then you will get only branches that are connected to the terminal bus of any generator, load, or switched shunt contained in the injection group. Device filtering can be used instead of more complicated Advanced Filters.

Advanced Filters Display

The Advanced Filters Display is a [Case Information Display](#) available under the folder *Case Information and Auxiliary* on the [Model Explorer](#). The purpose of this display is to list any [advanced filters](#) you have defined for the current case. Each record will list the name of the filter, the type of object the filter is for, filter logic, and whether or not a pre-filter was used.

Normally you will create advanced filters directly on a case information display that is showing a particular object type. However you can also choose **Insert** from the [Case Information Toolbar Records Menu](#) to create an advanced filter. You will then be prompted to choose the object type on a dialog before proceeding to the [Advanced Filters Dialog](#).

Normally you will delete advanced filters from the [Advanced Filters Dialog](#), however you can also choose **Delete** from the [Case Information Toolbar Records Menu](#) to delete an advanced filter. This will then remove the filter from any case info displays, [dynamic formatting definitions](#), [select by criteria definitions](#), or any other locations that are presently using this advanced filter.

The most common use of this display is to using the [Case Information Toolbar: Load Auxiliary Files Menu](#) **Save As** option to save a list of advanced filters to an auxiliary file. feature makes it easy to transfer defined Advanced Filters from one case to another.

Custom Expressions

Simulator allows you to define Custom Expressions that are functions of other fields. These Custom Expressions can then be shown as a column in a Case Information Display. To define custom expressions, right-click in a Case Information Display table and choose **Records > Define Expression** from the local menu or choose **Records > Define Expression** from the [case information toolbar](#). This brings up the Define Custom Expressions Dialog. When you bring up the dialog you will only see expressions that have been defined for the type of object shown on the Case Information Display (e.g. Bus, Generator, Interface, etc...).

To define a new custom Expression click **New**.

You can name the expression for easy identification in the list of fields for the object by filling in the **Expression Name** field.

To define the Custom Expression, first specify which fields you would like to use in the expression and assign them to the variables x1, x2, ..., x8. Fields can be chosen that apply to the type of object for which the expression is being defined or a [Model Expression](#) can be chosen. Model Expressions allow using either an expression that applies to model fields or simply a single model field. After fields or Model Expressions have been assigned to variables, type in the expression as a function of the variables x1, x2, ..., x8. For example

$x1 * \sin(x2) + \exp(-x5)$

or

$\tan(x1) + \text{abs}(x6) * 8 - 100$

For a complete list of functions and operators that are available to you, see [Functions and Operators Available](#).

Once the Custom Expression has been defined, you may add the expression to the column of the Case Information Display from which it was called by clicking on **Add Column...** For more information on how to add columns to a display see [Configuring the Case Information Displays](#).

Custom Expressions Display

To open the Custom Expressions Display, select **Case Information and Auxiliary > Custom Expressions** in the [Model Explorer](#). If you have defined any [custom expressions](#) for your case, you can see the list of those expressions in this display. The type of object to which the expression applies, the name of the expression, and the custom expression itself are given. The display also shows the values being represented by each of the variables in the custom expression.

Custom expressions cannot be inserted manually in this display, but they can be loaded from an [auxiliary file](#) by right-clicking on the grid and selecting **Load > Auxiliary File (any data)...** from the local menu. Conversely, you can save a list of custom expressions in a case to an auxiliary file by right-clicking on the grid and choosing **Save As > Auxiliary file**.

Custom String Expressions

Simulator allows you to define Custom String Expressions to generate a string from other fields. These Custom String Expressions can then be shown as a column in a Case Information Display. To define custom string expressions, right-click in a Case Information Display table and choose **Records > Define String Expression** from the local menu. This brings up the Define Custom String Expressions Dialog. When you bring up the dialog you will only see expressions that have been defined for the type of object shown on the Case Information Display (e.g. Bus, Generator, Interface, etc...).

To define a new custom Expression click **New**.

You can name the expression for easy identification in the list of fields for the object by filling in the **Expression Name** field.

To define the Custom Expression, first specify which fields you would like to use in the expression and assign them to the variables x1, x2, ..., x8. Fields can be chosen that apply to the type of object for which the expression is being defined or a [Model Expression](#) can be chosen. Model Expressions allow using either an expression that applies to model fields or simply a single model field.

The custom string expression is then defined in terms of the variables x1, x2, ..., x8. Different elements of the expression are concatenated with a + symbol. A number of different types of elements can be combined.

- Static strings can be specified inside of quotation marks
- Variables containing string data (such as Bus Name) may be added directly
- Variables containing numeric data may be formatted using the function `Str(variable, minlength, decimals)`. The *variable* parameter is the value to be formatted, the *minlength* parameter is the minimum number of characters for the result, and the *decimals* parameter specifies the number of characters to the right of the decimal point. If the *decimals* parameter is specified as a negative number, then all trailing zeros beyond the decimal are truncated.

For example

"This custom string expression displays the bus name (" + x1 + ") and per unit voltage (" + Str(x2, 3, 2) + ")"

For a complete list of functions and operators that are available to you for numeric expressions, see [Functions and Operators Available](#).

Once the Custom String Expression has been defined, you may add it to the column of the Case Information Display from which it was called by clicking on **Add Column...** For more information on how to add columns to a display see [Configuring the Case Information Displays](#).

Functions and Operators Available

Below is a list of functions and operators that are available for use in [Defining Custom Expressions](#):

Symbol	Equivalent	Description	Example
()		Prioritizes an expression	$5*(1+1) = 10$
!	FACT	Factorial	$5! = 120$ $\text{fact}(5) = 120$
%		Percentage	$35\% = 0.35$
^	**	Raised to the power of	$4^5 = 1024$
*		Multiply by	$3 * 6 = 18$
/		Divide by	$9 / 2 = 4.5$
\	DIV	Integer divide by	$9 \setminus 2 = 4$
MOD		Modulo (remainder)	$7 \bmod 4 = 3$
+		Add	$1 + 1 = 2$
-		Subtract	$9 - 5 = 4$
>		Greater than	$9 > 2 = 1$ * see note
<		Less than	$7 < 4 = 0$
==	=	Equal test	$5 == 4 = 0$
>=	=>	Greater or equal	$3 >= 3 = 1$
<=	=<	Less or equal	$\#h3E <= 9 = 0$
<>		Not equal	$\#b10101 <> 20 = 1$
NOT		Bitwise NOT	$\text{NOT}(15) = -16$
AND	&	Bitwise AND	$\#b101 \text{ AND } \#h1E = 4$
OR		Bitwise OR	$13 \text{ OR } 6 = 15$
XOR		Bitwise Exclusive OR	$9 \text{ XOR } 3 = 10$
EQV		Bitwise Equivalence	$6 \text{ EQV } 9 = -16$
IMP		Bitwise Implication	$1 \text{ IMP } 5 = -1$
IIF		If condition	$\text{IIf}(1+1=2, 4, 5) = 4$
MIN		Minimum value	$\text{min}(10, 3, 27, 15) = 3$

MAX		Maximum value	$\max(1,9)=9$ *see note
SIN		Sine	$\sin(\pi) = 0$ *see note
COS		Cosine	$\cos(\pi) = -1$
TAN		Tangent	$\tan(\pi) = 0$
ASIN		Arc sine	$\text{asin}(1) = 1.570$
ACOS		Arc cosine	$\text{acos}(-1) = 3.141$
ATAN	ATN	Arc tangent	$\text{atan}(0) = 0$
SEC		Secant	$\sec(0) = 1$
CSC		Cosecant	$\csc(1) = 1.18$
COT		Cotangent	$\cot(1) = 0.642$
SINH		Hyperbolic sine	$\sinh(3) = 10.01$
COSH		Hyperbolic cosine	$\cosh(2) = 3.76$
TANH		Hyperbolic tangent	$\tanh(1) = 0.76$
COTH		Hyperbolic cotangent	$\coth(1) = 1.31$
SECH		Hyperbolic secant	$\text{sech}(0) = 1$
CSCH		Hyperbolic cosecant	$\text{csch}(1) = 0.85$
ASINH		Hyperbolic arc sine	$\text{asinh}(2) = 1.44$
ACOSH		Hyperbolic arc cosine	$\text{acosh}(9) = 2.89$
ATANH		Hyperbolic arc tangent	$\text{atanh}(.1) = 0.10$
ACOTH		Hyperbolic arc cotangent	$\text{acoth}(7) = 0.14$
ASECH		Hyperbolic arc secant	$\text{asech}(.3) = 1.87$
ACSCH		Hyperbolic arc cosecant	$\text{acsch}(2) = 0.48$
ABS		Absolute value	$\text{abs}(-8) = 8$
EXP		e to the power of	$\exp(3) = 20.08$
EXP2		2 to the power of	$\exp2(3) = 8$
EXP10		10 to the power of	$\exp10(3) = 1000$
LOG	LN	Natural log	$\log(16) = 2.77$

LOG2		Log base 2	$\log_2(8) = 3$
LOG10		Log base 10	$\log_{10}(100) = 2$
CEIL		Round up	$\text{ceil}(6.2) = 7$
RND		Random number	$\text{rnd}(1) = .969$
INT		Truncate to an integer	$\text{int}(6.8) = 6$
SGN	SIGN	Sign of expression (-1, 0, or 1)	$\text{sgn}(-9) = -1$
SQR	SQRT	Square root	$\text{sqr}(64) = 8$

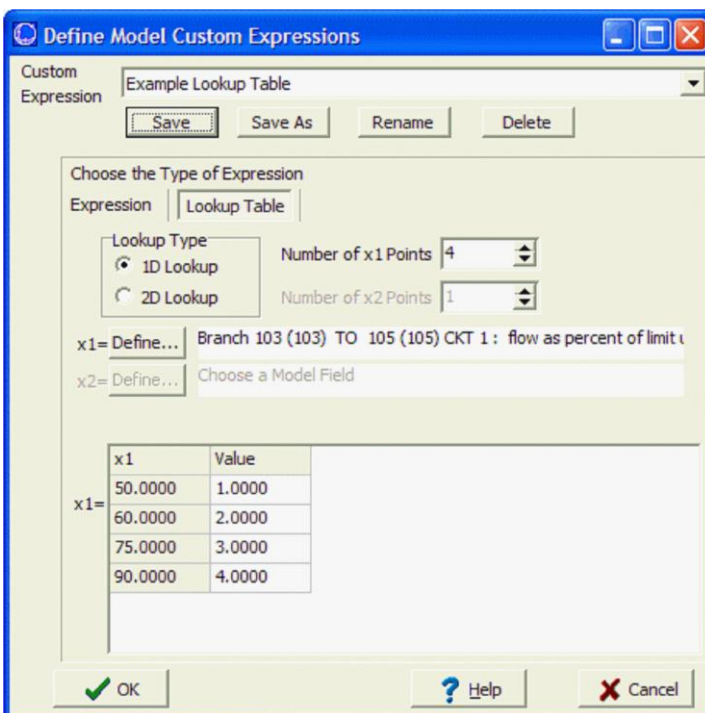
Model Expressions

Model Expressions can be accessed and created by choosing **Case Information and Auxiliary > Custom Model Expressions** in the [Model Explorer](#). To open the Define Model Custom Expressions dialog, right-click and choose **Insert**, when creating a new expression, or **Show Dialog** to view an existing expression.

The **Save**, **Save As**, **Rename**, and **Delete** buttons and the drop-down list of expressions at the top of the dialog allow for the additional of new expressions, modification of existing expressions, and removal of existing expressions. There are two types of Model Expressions: an expression and a lookup table.

Lookup Tables

The dialog as configured when creating a lookup table appears as follows:



A lookup table may be either one or two-dimensional by specifying the **Lookup Type**. The **Number of Points** must be set for each dimension. This will determine the size of the table. A Model Field must also be selected for each dimension. Clicking the **Define** button next to either the **x1** or **x2** field opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements behaves the same as the [Find Dialogs](#) found throughout Simulator. After selecting the element, choose the field associated with this element that will be used in the Lookup Table. Click **OK** to accept the element and field selected and close the dialog.

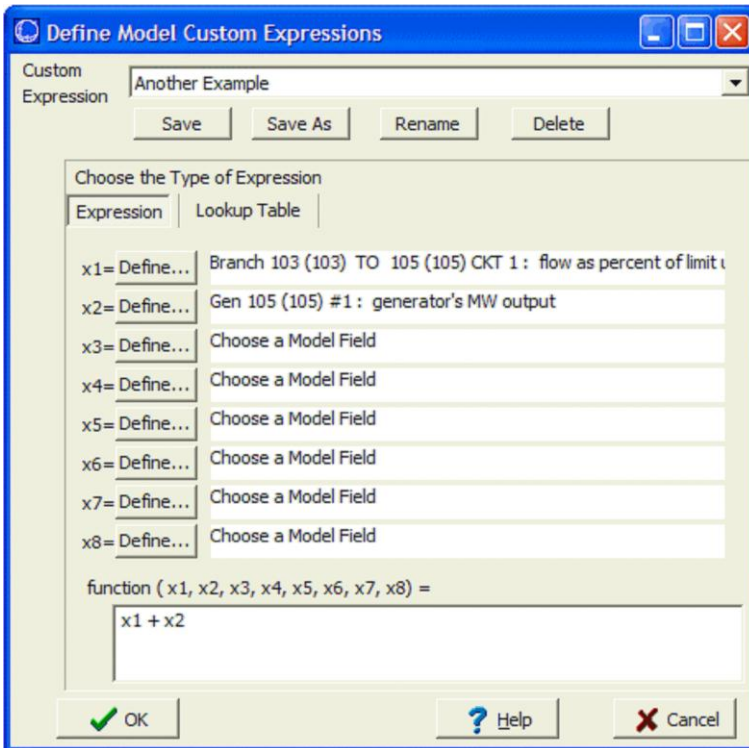
Fill in the Lookup Table with the desired values. If selecting a one-dimensional table, there will be two columns, one containing the values for the **x1** Model Field and the other containing the Values to return from the Lookup Table when the Model Field is at the defined value. There will be one header row plus as many additional rows as defined in the **Number of x1 Points**. If selecting a two-dimensional table, there will be one column for the **x1** Model Field and as many other columns as selected for the **Number of x2 Points**. There will be one header row for defining the **x2** values plus as many additional rows as defined in the **Number of x1 Points**. The cells under the first row

and to the right of the first column are used for defining the Values to return from the Lookup Table when the Model Fields meet the defined values.

If selecting a one-dimensional table, the returned Value is determined based on the current value of the **x1** Model Field. The returned value is the Value corresponding with the **x1** value that is less than or equal to the current value of the Model Field. If the current value of the Model Field is less than the smallest value of **x1** in the table, then the Value returned corresponds to the smallest **x1** value. If selecting a two-dimensional table, the same rules apply except that the Value returned corresponds to the current value of the **x1** AND **x2** Model Fields and where these intersect in the table.

Expressions

The dialog as configured when creating an expression appears below:



To define the Model Expression as an expression, specify the Model Fields that the expression should be a function of by clicking the **Define** button next to the appropriate variable. Clicking the **Define** button opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements of a particular type can be searched, filtered, and sorted to make finding an element easier. After selecting the element, choose the field associated with this element that will be used in the expression. Click **OK** to accept the element and field selected and close the dialog. After the Model Fields have been defined, enter the function description. See [Custom Expressions](#) for more information about defining expressions.

After model expressions have been defined, new options will appear on the [Contingency Element Dialog](#) and on the [Advanced Filter Dialog](#).

Normally, you may only enter a constant for the comparison value on the [Advanced Filter Dialog](#). However, if you select **Enable Field to Field Comparisons**, the advanced filter dialog will feature a drop-down from which you can choose *Expression*. After choosing *Expression*, you may select the name of the model expression, or click the **Find** button to search for the name.

Advanced Filters for Substation

Filter Name:

Save Save As Rename Delete

☐ Pre-filter using Area/Zone/Owner Filters

Logical Comparison: ☒ AND ☐ OR ☐ Not AND ☐ Not OR

☐ Limit Fields to Columns ☒ Show All Fields

Condition 1: ☒ Find... ☐ ABS

☐ Use Another Filter Find

Add >> Delete ...

☒ Enable Field to Field Comparisons

Filter Remove Help Cancel

Similarly, normally you may only enter a *Constant* or a *Field* for the *Move*, *Set To*, or *Change By* **Action Types** on the [Contingency Element Dialog](#). However, now you will have the option to choose *Model Expression*. To change the type of the **Amount**, choose *Model Expression* from the second drop-down box. After doing this, enter or choose the name of the model expression in the first **Amount** drop-down box, or click on the **Find...** button to search for a name.

Contingency Element Dialog

Element Type: ☐ Branch ☒ Generator ☐ Load ☐ Switched Shunt ☐ Bus ☐ Interface ☐ Injection Group ☐ Series Capacitor ☐ DC Line ☐ Phase Shifter ☐ 3-Winding Transformer ☐ Solve Power Flow ☐ Contingency Block

Choose the Element: ☐ Sort by Name ☒ Sort by Number

Define Filter ☐ Use Area/Zone Filters

1

1 (Bus 1) [138 KV]
2 (Bus 2) [138 KV]
4 (Bus 4) [138 KV]
6 (Bus 6) [138 KV]
7 (Bus 7) [138 KV]

Action Type: ☐ Open ☐ Close ☐ Move ☐ Set To ☒ Change By

Amount: Find...

Make-up Power Sources ...

in: ☐ MW (const pf) ☒ Percent ☐ MW ☐ MVAR ☐ Setpoint Voltage

Model Criteria: Add

Comment:

OK Delete Cancel Help

Once a Model Expression has been saved, it may be referenced directly in a Case Information Display or AUX file using [special notation](#).

Find Dialog Basics

Many times when working with large load flow cases, it can be somewhat difficult to locate devices in the [case information displays](#) regarding a specific device. Simulator has many tools to facilitate filtering data, such as the [Area/Zone/Owner filters](#) and the [Advanced Filtering](#) tool. Even with these helpful tools, finding a device can still be hampered when the bus number or exact spelling of the bus name are not known.

To facilitate locating devices in Simulator, you can use the Find tool to use Simulator's advanced search engine for finding the device(s) you are looking for. The Find tool is available from the popup menus of almost all [case information displays](#), as well as several of the various information dialogs in Simulator. Anywhere you see a button or menu option labeled **Find...** you can open the advanced search tool.

Once the Find dialog has been opened, the dialog will automatically adjust to suit the type of device you are searching for. The caption of the dialog should reflect the type of object the dialog is currently attuned to locate. For most devices, such as buses, generators, loads, etc., there will be one list displayed at the bottom of the dialog containing numbers and names of the type of device you are searching for. In some instances, mostly when searching for branch-type devices, the bottom panel is split with a second list is displayed in the right side panel. This list is used to display the possible connections of the bus selected in the first list. For example, if bus number one is selected and it has connections to bus two and bus three, the second list will display the information for bus two and bus three. Thus you can search for a bus in the first list, then choose from the possible connections in the second list to get a specific branch from the list.

Despite what type of device you are trying to find, the first few options and buttons on the dialog will be the same. **Sort by Name** and **Sort by Number** allow you to choose how you wish to find a device in the list. If you know the bus number you are looking for, choose **Sort by Number**. If you know the name, or at least part of the name, that you are looking for, then choose **Sort by Name**. The list (or first list for branches) will be sorted accordingly.

If you wish to narrow down the list of devices to search through, you have a couple of options for filtering the list before searching through it. First you can make use of the traditional [Area/Zone/Owner filters](#) by clicking in the associated check box. If you need to set more specific conditions for filtering the list, you can instead click the **Define Filter** button to set up an [advanced filter](#). Either one will reduce the number of devices in the list for the search.

Once you have the list set up for your search, you can type in the number or name you wish to find in the text box. If you do not know the exact number or name, you can use wildcards to facilitate the search and find all possible matches for a set of characters or numbers. You can use a question mark (?) to represent a single character wildcard, or an asterisk (*) to represent a multiple character wildcard.

For example, if you want to find bus number 10005, but all you know is the first four digits are 1000, you can type in *1000*, and Simulator will search until it finds the first number that contains those four digits. You can then use the **Search Next** button (note that pressing **Enter** is the same as clicking **Search Next**) to find the next number containing the four numbers, and so on. The same goes for searching by name. If you are looking for bus ACEONE, but all you know is the name contains the string ACE, then you can type in *ACE* and then keep pressing **Enter** until you find the bus named ACEONE. By using the beginning and ending *, we would also find elements such as NEWACE, because the double * looks for strings that contain ACE anywhere in the string. Note that if you know the first few letters (or numbers), you can narrow down the number of elements found from the search by eliminating the first * from the search string. For example, to find bus ACEONE, we could have instead used ACE*, and this would have gone through all matches that started with ACE, ignoring other elements such as NEWACE. You can also use wildcards in the middle of a string, such as AC*NE, and Simulator will find any name that starts with AC and ends with NE.

Once you have entered a wildcard search, you can also click the **Search All** button. This will essentially bring up the same Find dialog again, but with the choices narrowed to those that meet your wildcard search.

Search for Text Dialog

This dialog allows you to search for specific text in a case information display. Specify the text you want to search in **Search for** edit box. Clicking on **Search Next** will take you to the next field whose content matches with the text specified. The search can be made **By Rows**, in which the text is searched first in all the fields of a record, before searching in the next record. If the search is made **By Columns** then the text is searched first in all the fields of a column, before searching the text in the next column. If the option **Match case** is checked, the search for text will be case-sensitive. The option **Find entire cells only** will take you only to fields whose entire content matches completely with the text you are searching for.

When checking the box **Search Ignore Filters**, Simulator will look through all the records of the type on your present case information display, but will ignore any [Area/Zone/Owner Filtering](#) and any [Advanced Filtering](#) specified with that case information display. If text is found which matches your wildcard search, but is for a record presently filtered out, then you will be prompted to remove the filtering so that the record can be displayed.

When checking the box **Search All Fields**, Simulator will look through every available field which can be displayed for the type of records presently represented on your case information display. If text is found which matches your wildcard search, but is for a field that is not presently shown, then you will be prompted to add the field to the columns that are shown.

Model Conditions Display and Dialog

Model Conditions are a type of Model Criteria. Model Criteria represent Boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional based on the state of the power system. To see a list of Model Conditions, choose **Case Information and Auxiliary > Model Conditions**.

A model condition contains two parts: a power system element and an [advanced filter](#). The model condition will then return true or false depending on the result of applying the advanced filter to the power system element specified. Model conditions can also be used as part of a [Model Filter](#).

The Model Conditions Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model condition, right-click on the display and choose **Delete**. To insert a new model condition, right-click on the display and choose **Insert**. This brings up the Model Conditions Dialog that can be used to create, delete, and modify Model Conditions.

Model Conditions Dialog

The Model Conditions Dialog has three sections. The top section provides the ability to **Save**, **Save As**, **Rename** and **Delete** model conditions. To choose a different Model Condition, click on the down arrow next to the Model Condition name.

The middle section provides a location to specify what power system element this Model Condition is related to. On the left is a list of **Element Types** that are available. When clicking on one of these types, the right portion of the dialog will provide you with a list of the elements of this type. This list is a familiar [Find Dialog](#) and provides you the ability to search for the element you are interested in.

The bottom section provides a location to specify an [Advanced Filter](#) which you would like to have applied to the power system element chosen in the middle section. The bottom section of the dialog behaves identically to the [Advanced Filtering Dialog](#). If an Advanced Filter exists for the type of element for which you are creating a Model Condition, you can click the **Set Filter Same As** button to choose this filter. This will then set the parameters of the Model Condition to be the same as the Advanced Filter.

Model Filters Display and Dialog

Model Filters are a type of Model Criteria. Model Criteria represent boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional on the state of the power system. To see a list of Model Filters, choose **Case Information and Auxiliary > Model Filters** in the [Model Explorer](#).

A model filter contains a list of [Model Conditions](#) and/or other Model Filters and a boolean operator to apply to these conditions. The model filter then returns true or false depending on the result of applying the boolean operator to the boolean results of the conditions. A **NOT** operator can be applied to each condition so that the boolean opposite of the condition result will be used when evaluating the filter.

The Model Filters Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model filter, right-click on the display and choose **Delete**. To insert a new model filter, right-click on the display and choose **Insert**. This brings up the Model Filters Dialog that can be used to create, delete, and modify Model Filters.

Model Filters Dialog

The Model Filters dialog behaves in a manner very similar to the [Advanced Filtering Dialog](#). To choose conditions for the model filter, click the down arrows next to the conditions list. The list will either contain Model Conditions or Model Filters depending on the option to display either **Condition** or **Filter**. You may also click the **Find...** button to the left of the list. To insert or delete conditions from the model filter, click the **Add>>** or **Delete...** button. When specifying more than one condition, choose the **Logical Comparison** that will be used to compare the conditions. Individually each condition can have a NOT operator applied to it by checking the **NOT** checkbox next to the condition. Finally, the top of the dialog provides the ability to **Save**, **Save As**, **Rename** and **Delete** model filters. To choose a different Model Filter, use click on the down arrow next to the Model Filter name.

To modify the model conditions in the case, click the **Modify Model Conditions Dialog**. This will open the [Model Conditions Dialog](#).

Select [View Filter Logic](#) to show a graphical display to visualize the logic diagram for a Model Filter.

Model Filters View Filter Logic Graphical Display

The **Model Filter Logic Display** feature provides a graphical display which allows you to quickly browse information about a [Model Filter](#) and the corresponding [Model Conditions](#). The model filter logic displays enable convenient Model Filter-by-Model Filter navigation through the complete list of model filters. From the model filter logic display, you can find out a model filter name and Logical Comparison object, and model conditions names that are part of the particular model filter. Moreover you can find out all information associated with the model filter or the model conditions by pressing right click with the mouse and directly invoking their associated information dialogs. The advantage of the model filter logic displays, is that they are auto-created logic diagrams.

Along the top of the model filter view display resides a panel of controls. To move the model filter logic view to a particular model filter, use the up and down arrow to move to the desired the model filter in the list. . If you are navigating and want to return to a previously open model filter press **History** to bring up a menu list of the previously open model filters. The menu will show a list of the last model filters which have been visited in a model filter view.

Below this top panel sits the actual model filter display. The model filter you have chosen to inspect, is represented as the last model filter to the right of the display. Notice that from this model filter there are also a connection to other model filters that are in a column. They represent the model filters in which the selected model filter is used as a model condition . When you drag the mouse over one of the neighboring model filters symbols, it turns into a pointing finger, then by clicking the left mouse button when the mouse cursor is in this shape redefines the target model filter to be the model filter whose symbol you just clicked. The model filter view display is redrawn to show the same sort of display for the newly chosen target. Right-clicking on the model filter view display's background will generate the local menu.

The model filter background color can be red or green. Green means that the model criteria is satisfied and the evaluation of the model filter is true. Red means that the evaluation is false. The model filter objects also can have a circle entering to the model filter. This means that the connecting model filter or model condition will be negated. For example, if the model condition or filter entering another model filter model criteria is true it will evaluated as false in the entering model filter. Repeated colors of the Model Filter Names and Conditions represent that the particular Model Filter or Condition is used more than once in the Model Filter Display.

The model filter view display can be generated using any of the following methods:

- Right click on the model filter of interest on the [Model Explorer](#) on the model filter [Case Information](#) of the contingency analysis folder. This will bring the local menu and the display can be opened by selecting "**Insert Filter Logic...**"
- From the [Contingency Analysis Tool Dialog](#), Go to the [Contingency Definitions](#) inside the Options ribbon. In the model filters definitions case information repeat the previously explained process.
- Pressing the "**View Filter Logic**" button in the [Model Filter Display](#) Dialog.

Entering a Range of Numbers

On a number of displays it is often convenient to enter a group of numbers, including ranges. Examples include entering buses or areas to scale on the [Scaling Display](#), or buses on the [Quick Power Flow List](#). The format for this field is to enter individual numbers separated by commas, and/or ranges with a dash between the beginning of the range and the end of the range.

For example the entry

1-5,21,23-25

corresponds to numbers 1 through 5, 21 and 23 through 25.

Referencing Model Expressions

A saved [Model Expression](#) may be referenced directly in a Case Information Display or AUX file using special notation. A string of the format "&*ModExpName*:*digits*:*decimals*" will resolve to the result of the model expression named by *ModExpName*, with *digits* specifying the minimum number of digits to display, and *decimals* specifying the number of digits to show to the right of the decimal.

Note that the link to the Model Expression is not maintained after the initial entry. If the Model Expression is modified, or any of the relevant values change, the value in the Case Information Display will not update unless the reference is re-entered manually or the AUX file is re-loaded.

Example:

&NetGeneration:5:2

Copying Simulator Data to and from Other Applications

You may sometimes find it useful to copy data from Simulator to other applications such as a word processor or spreadsheet program. Alternatively, you may want to copy data from other applications into Simulator. Simulator's [case information displays](#) provide a convenient way to accomplish this. In particular, the **Copy All**, **Copy Selection**, and **Paste** options from the local menu or the [Case Information Toolbar: Copy, Paste and Send Menu](#) allow Simulator to pass data with other applications through the windows clipboard.

To copy a selection of data from a case information display to another application, first select the range of cells to copy from the case information display. Then, right-click on the case information display to display its local menu, and choose **Copy Selection**. To copy the entire content of a case information display to another application, follow the same procedure, except choose **Copy All** from the local menu instead of **Copy Selection**. Switch to the application that will serve as destination for the data and use that application's **Paste** command to finish copying the selected Simulator data to that application. Note that not only is the data copied, but by default so are the data headings. The data headings are very important to maintain if you are planning on copying data from a spreadsheet back into a Simulator case information display. Also note that Simulator copies the text to the windows clipboard with tabs delimiting each cell.

Note: Be aware that most programs have limitations on the amount of information you can paste. For example, some spreadsheet programs only allow up to 256 columns and 65,000 rows of information. Some power system information, particularly the Ybus or Jacobian matrices, can easily exceed these limitations.

To paste data from another application into a case-information display, select the data in the other application and use that application's Copy command. In order to paste back into Simulator, you must have the record type and data headings selected with the columns of data. The record type (or object name) must match the case information display you are attempting to paste into, and the data headings (or variable names) tell Simulator which columns of data you are pasting. In order for the data to be pasted to the correct records, you must include the [key field](#) columns in the data to be pasted into Simulator. Also if you are attempting to create new data records by pasting back into Simulator then you must include the [required field](#) columns as well. If the object type is not valid then the **Paste** option will be disabled and the caption will read "object type invalid". If the key fields are missing, then the **Paste** option will be disabled and the caption will read "key fields missing".

It is not necessary to copy and paste all columns or rows of data back into Simulator, as long as the column headings match a valid heading for the case information display. Simulator will skip the data under any column headings that are unrecognized. If you wish to copy data from a spreadsheet into a Simulator display, you must make sure they are ordered in such a way that all the information you wish to paste can be grouped and copied as one block of data from the spreadsheet. Once you have copied all the information from the spreadsheet you wish to transfer, switch to Simulator, open the case information display in which to paste the data, and select **Paste** from its local menu.

Note: You can only paste values into the case information displays if the values are enterable on the display (shown blue by default). Also, be careful about pasting redundant data. For example, in the Bus Records display both voltage in *per unit* and voltage in *kV* are enterable, but they specify the same information. Make sure you only copy ONE of these columns into Simulator. Otherwise you may not get what you expect. Simulator will paste the value in twice, and whatever value was pasted 2nd will be used.

Save Case Information Data

The Save Data Dialog gives you quick access to saving data from certain [Case Information Displays](#) in a PowerWorld Simulator [Auxiliary](#) File.

Specify the name of the auxiliary file in which to save the data records in the text box labeled **Name of File to Save**. Instead of typing the name of the file by hand, you can press the **Browse** button to locate it. Then, indicate whether you want the objects whose data you are writing to the file to be identified by number or by name. Finally, to ensure that only the records currently listed in the [Case Information Display](#) are written to the auxiliary file, check the **Save Only Records Listed in this Display** checkbox. Otherwise, data for all such objects in the entire system will be saved to the file.

When you have finished setting these options, click **OK**. If you changed your mind and do not want to save the data to a file, click **Cancel**.

Key Fields

Key fields are necessary fields when attempting to paste data into a [Case Information](#) display from another data application. Other fields are also considered [required fields](#) for the purpose of creating new objects in Simulator by pasting or loading information from another source, such as Excel or a [PowerWorld Auxiliary](#) file. The key fields are easily identified from a few different locations.

Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

The following table lists some of the more commonly accessed key fields needed when attempting to Paste data into existing objects in a Case Information Displays from another data application.

Type of Data	Data Description	Necessary Key Columns for Paste		
Area	Area Records	Area Num		
Bus	Bus Records	Number		
DC Line	DC Line Records	Rectifier Number	Inv Number	Num
Gen	Gen Records	Number	ID	
Interface	Interface Records	Name		
Line/Transformers	Line Records	From Number	To Number	Circuit
Load	Load Records	Number	ID	
Substations	Substation Records	Sub Num		

Switched Shunt	Switched Shunt Records	Number	ID
Zone	Zone Records	Zone Num	

Required Fields

Required fields are necessary fields when attempting to create new objects by pasting data into a [Case Information Display](#) or when reading data from an [PowerWorld Auxiliary](#). These fields are in addition to the [key fields](#) which identify the object.

There are two purposes of the required fields. First there are some fields for which an appropriate default value is not apparent thus by requiring these fields it forces the specification of the value. Second, when pasting data into Simulator it is possible that the record will not exist in the case. Simulator will then determine whether to create these records by looking to see if the required fields are specified. By doing this, it prevents the accidental addition of records. For instance you may have an Auxiliary File that sets the Monitor field of all branches in your case to Yes or No. If Simulator runs across a branch between two buses that exist, but whose circuit ID does not exist, then Simulator will just skip this line from the Auxiliary File. It will not be possible to accidentally create a bunch of new transmission lines with default impedance parameters because the R and X impedance parameters are required.

The required and key fields are easily identified from a few different locations.

Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

An example of required fields for an object is for a Branch object. For a branch object, the key fields are the from bus number, to bus number and the circuit ID. Required fields are the Impedance Values R, X, and B and the first three ratings A, B, and C. Without specifying values for these fields you can not create a new branch.

Contour Column Dialog

The Contour Column Dialog allows you to contour case information display columns.

The Contour Column Dialog has two tabs:

[Contour Type](#)

[Custom Color Map](#)

Contour Column Type

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

Reverse Color Map Colors

Check this box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use cell values directly

Select this option to use the cell values to do the contouring.

Use the specified field below

Select this option to use a different field value to do the contouring.

Value

Select the quantity to use in the contouring from the Value dropdown box or click the **Find Value** button to find the desired field.

Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

Title

Title for the color key.

Entry Labels

Units of the contoured value displayed on the color key.

Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

Scalar

Multiplication factor that can be applied to the values when drawing the color key.

Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent.

Use absolute value

Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

Values

These values along with the color map define how to convert your values into a color for the contour. The values are:

Maximum The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

Grid Metrics Dialog

The grid metrics dialog allows you to determine the metrics for a set of columns. The metrics determined include: Sum, Average, Variance, Standard Deviation, Maximum, Minimum, Total Items, and Total Non-Zero Items.

By default the dialog is set to determine the metrics for the whole column of the case information display where the option has been selected. However, the selected record set can be modified. The **Start Column** and **End Column** values specify what columns to include in determining the metrics. The **Start Row** and **End Row** values specify what records to include when computing the metrics. If the option **Treat blank cells as zero** is checked, then any blank cells will be considered as zero when computing the metrics. Instead, if the option **Ignore blank cells for calculation** is checked, any blank cell will be taken out of the metrics calculation. Check **Use Absolute Values** to use the absolute values of field values for the metrics computation. Finally, click **Update Metrics** to determine the metrics with the new set of options.

Auxiliary File Export Format Description (For both Display and Power System)

Definition of an Export Format Description

An export format description consists of the following items:

1. A set of object types
2. The fields and subdata to be exported for each object type, including how each field should be formatted when written (e.g., number of digits, number of decimal places)
3. The format (comma-delimited, space-delimited, etc.) that should be used when writing the object data to disk.

For instance, an export format description might have two object types defined—buses and generators—with indications to export the bus numbers for each of these objects and to write this information to a space-delimited .AUX file.

Types of Export Format Descriptions

Export format descriptions can be defined for both the Display Objects and the Power System Objects,

- Display objects, e.g., background lines, pie charts, displayed fields, etc. Export format descriptions for display objects are defined by selecting Display Objects Export Format Description... on the [Onelines](#) ribbon tab, under the List Display Menu on the **Active** ribbon group.
- Power system objects, e.g., buses, loads, generators, etc. Export format descriptions for power system objects are defined by selecting Power System Objects Export Format Description from the [Case Information](#) ribbon tab on the **Case Data** ribbon group.

The dialogs used to define both types of export format description are very similar; any differences between the two dialogs will be clearly noted below.

Export Format Description Dialog

Clicking on **Display Objects Export Format Description** or **Power System Objects Export Format Description**, opens the Export Format Description Dialog. There are several buttons at the bottom of the dialog:

- **Create Format for Complete Case** : sets the format for the AUX file to be created to be all necessary fields to replicate the load flow case. This is essentially the same as choosing to save the case as an auxiliary file when choosing "Save Case As" from the Application menu and setting the "Save as type" parameter to "PowerWorld Auxiliary Data".
- **OK** : save the current settings to the selected export format description, and close the dialog
- **Create AUX File with Specified Format** : writes data to a file using the current settings. A dialog pops up to select the filename for storing the data. If the file already exists, the data is appended; otherwise, a new file will be created
- **Cancel** : do not save the current settings and close the dialog
- **Help** : open the help associated with the dialog .

The other parts of this dialog box are discussed in detail below

Format Name Section

The uppermost part of the dialog box is used to manage the set of export descriptions. Once one export description has been saved (by clicking Save or Save As, after defining some object types), a drop-down box will appear which allows selection of an export format description from the set of export format descriptions saved with the case: The buttons underneath the drop-down box have the following functions:

- **New** : clear the rest of the form in order to define a new export format description
- **Save** : save the information in the rest of the dialog box to the export format description currently selected in the drop-down box. If no export format description is selected, this is equivalent to clicking Save As.
- **Save As** : save the information in the rest of the dialog box to a new export format description; a dialog will pop-up asking for the name of the new export format description.
- **Rename** : rename the currently selected export format description
- **Delete** : delete the currently selected export format description

The list of export format descriptions for display objects and power system objects is kept separate, so the drop-down box will only contain export format descriptions defined for the object type selected from the menu (Display Objects Export Format Description Dialog or Power System Objects Export Format Description Dialog).

File Type and Field Defaults

The next section of the dialog box is used to define the format to save the data, and the default formatting of any numeric fields that are to be saved. Currently, there are 3 options for the file type:

- **AUX (space)** : output the data using a space-delimited .aux file format
- **AUX (,)** : output the data using a comma-delimited .aux file format
- **CSV** : output the data to a comma-separated-values (CSV) file. This type of file can be easily imported into any spreadsheet or database software.

In the **Defaults for Exporting Fields** section, the up/down arrows can be used to change the default number of total digits and digits to the right of the decimal place used in exporting numeric values. These values may be overridden in the Select Fields form for individual fields.

Object Type and Filter Method Section

The two-column section on the left side of the dialog box is used to define object types and a filter for each object type.

Object Type

The first column is used to specify which object types are to be exported. To insert a new object type (corresponding to a new row) into the export format description, there are two options—either click the New button above the **Object Type** column heading, or right click on a row and select **Insert**. Upon doing this, a dialog will pop up to choose an object type:

Type into the text box to search for a particular object type. The list of object types depends on the type of export description being defined. When defining display object export descriptions, only display object types will show up in this list; similarly, when defining power system export descriptions, only power system objects will show up.

Clicking **Choose** will insert the object type into the list of object types to be exported.

Double-clicking an existing object type will pop-up this same dialog box, allowing the object type to be changed. Doing so will empty the exported fields, subdata, and filter method.

Filter Method

The second column is used to define an object filter. The default behavior is to export all objects without any filtering, but this can be changed by double-clicking on the filter description. Upon double-clicking on the filter description, a dialog will appear allowing you to choose the filter. The options available for determining which objects are exported are as follows

- **All** : export all objects, disregarding any filters defined in the current case
- **Use Area/Zone/Owner Filter** : only export those objects that satisfy the currently defined area/zone/owner filter. Clicking on **Select Area/Zone** will pop up the dialog box used to define area/zone/owner filters.
- **Use Selected** : only export those objects that have the Selected field set to Yes. Clicking on **Select** pop up the dialog box used to set the Selected flag for each of the objects in the case. Note that this particular option does not exist for display objects, because display objects do not currently support the Selected field.
- **Meets Filter** : only export those objects that meet an advanced filter, which can be either selected from the drop-down box or defined using the **Define Filter** button.

Fields And Subdata To Be Exported

After specifying an object type (and, optionally, a filter), selecting the row on the left side of the dialog box will allow you to specify the fields and subdata to be exported.

Fields To Be Exported

By default, no fields are selected for export. To define fields for export, click on the **Modify** button to the right of the Fields heading the **Select Fields dialog** box will popup. This dialog is used to specify which fields are to be exported, along with any field-specific formatting of numerical values . Generally the dialog behaves identically to the [Configuring Case Information Displays Dialog](#), so for more detailed help see that topic. There are parts of the dialog that behave differently however, and these are described as follows.

- **Total Digits / Dec Places** : these two boxes are used to set the number of total digits and the number of decimal places to use when writing the data for the fields current selected in the Selected Fields section. Leaving these two boxes blank means that the default number of digits and decimal places should be used. Also, if multiple fields are selected that have different settings for the number of digits or number of decimal places, these boxes will be greyed out, but still enterable. Once entering a value into either box, all selected fields will then have the same setting, and the box will no longer be greyed out.

Subdata To Be Exported

By default, no subdata is selected to be exported. Also, many objects (e.g., Bus and Area) do not have subdata associated with them, so the **Modify** button will be greyed out. If an object type does have subdata, then the SubData definition section will be greyed out.

To add subdata that should be exported, click the **Modify** button to the right of the SubData heading. This brings up the Select SubData dialog. A description of the options/settings on this dialog follows.

- **Available SubData** : lists the SubData that exist for this object type. If Exclude subdata already selected is checked, then only SubData that is not already in the Selected SubData section will show up in this list.
- **Exclude subdata already selected** : if checked, then the Available SubData section will only contain items that are not already in the Selected SubData section
- **Selected SubData** : lists the SubData to be exported for this object type
- **Add ->** : set the currently selected SubData in the Available SubData section to be exported
- **<- Remove** : set the currently selected SubData in the Selected SubData section to not be exported
- **OK** : save the settings and close the dialog
- **Save** : save the settings, but leave the dialog open
- **Cancel** : do not save the settings and close the dialog
- **Help** : click on this to open the help associated with the Select Fields dialog

Geographic Data View

Geographic Data Views allow the quick creation and formatting of graphical representations of buses, generators, loads, switched shunts, transmission lines, substations, areas, zones, super areas, and injection groups. Latitude and longitude coordinates specified with buses and substations are used to place objects geographically on a display. The latitude and longitude coordinates must be specified in order to use this feature. Object attributes such as color, size, rotation rate, rotation angle, and visibility can easily be formatted based on the associated data object field values.

To create a Geographic Data View, open a case information display for the desired element type, select the elements for which to add geographic representations, and either right-click on the object grid and choose **Geographic Data View > Geographic Data View** from the local menu or choose **Geo > Geographic Data View** from the [case information toolbar](#). Use the **Select Column**, then **Geographic Data View** option to select an entire column of elements. The column in which the elements are selected determines the initial selection of the field to use for [attribute settings](#). The Geographic Data View Customization dialog will open.

Geographic Data View Customization Dialog

The screenshot shows the 'Geographic Data View Customization Dialog' with the following sections:

- Online for Geographic Data Addition:** Includes radio buttons for 'Use Already Open Online' (selected) and 'Open a Online'. The 'Use Already Open Online' section has a dropdown menu showing 'GEODATAFORM_Simulation_1' and a 'Browse...' button. There are buttons for 'Create Online with Geographic Borders' and 'Create Empty Online'.
- Class and Field Display Option Sets:** Includes a 'Class Type' dropdown set to 'Bus' and a 'Style Name' dropdown set to 'Bus_BusNum'. There are buttons for 'Save', 'Save As', 'Rename', 'Delete', 'Save to Auxiliary', and 'Load from Auxiliary'.
- General Display Options:** This section is active and contains:
 - Style:** Radio buttons for 'Rectangle' (selected), 'Rounded Rectangle', 'Oval', 'Triangle', and 'Kite'.
 - Object ID:** Radio buttons for 'Show Object ID by' with options 'Name' (selected), 'Number', 'Name (Number)', 'Number (Name)', and 'None'. There is also a checkbox for 'Prefix ID with Class Type'.
 - Selection Options:** Includes a checkbox for 'Can Be Selected' and a 'Line Thickness' spinner set to 1. The 'Line Color' is shown as a green square.
 - Animation Control:** Includes a checkbox for 'Enable Animation' and an 'Interval (Milli-seconds)' spinner set to 100.

At the bottom, there are buttons for 'Create Geo Data View Display and Close Dialog', 'Cancel', and 'Help'.

Online for Geographic Data Addition

When creating a new geographic data view, the oneline to which to add the geographic data view objects must be set. Choose either to **Use Already Open Oneline** and then select a oneline from the dropdown list of available onelines or choose to **Open a Oneline** and then either type in the name of a PWD file or click Browse to select a one.

Because geographic data views are dependent on having geographic latitude and longitude information available, the oneline that is selected must have a valid map projection in use. Only those open onelines that currently have a [map projection](#) in use will be listed in the dropdown box and any oneline opened from a file will be checked for a valid map projection. If a valid map projection does not exist, new geographic data view objects will not be added to the oneline.

Create Oneline with Geographic Borders

Click this button to create a new oneline diagram with first being prompted to add geographic borders. A default name will be assigned to the new oneline and that oneline will be added as the selected oneline in the Use Already Open Oneline dropdown.

Setting Attributes for Geographic Data View Display Objects

Before adding new geographic data view objects, the attributes associated with the objects should be set. These attributes are stored in [Geographic Data View Styles](#).

Create Empty Oneline

Click this button to create a new oneline diagram. A default name will be assigned to the new oneline and that oneline will be added as the selected oneline in the Use Already Open Oneline dropdown.

Create Geo Data View Display and Close Dialog

Click this button to accept all of the option settings, add new geographic data view objects to the selected oneline, and then close the dialog.

Cancel

Click this button to close the dialog without adding and new geographic data view objects.

Geographic Data View Styles: Fields and Attributes

The Field and Attributes options associated with Geographic Data View Styles allow the formatting of the geographic data view display objects depending on the values of fields of their linked data objects.

The screenshot shows the 'Fields and Attributes' tab of a dialog box. It is divided into two main sections. The left section contains settings for various attributes: Line Thickness (1), Line Color (black), Fill Color (cyan, with 'Use Fill' checked), Total Area (Width: 9.55, Size to Width Ratio: 0.500, with 'Use Auto Width of Objects' checked), Rotation Angle (0.0 Degrees), Rotation Rate (0.0 Hz), and Visibility (All Visible selected). Each attribute has a 'Use' checkbox and a 'Field Lookup' dropdown menu with a 'Find...' button. The right section is titled 'Fill Color Lookup' and includes 'Field Set Metrics' (Min: 1.0000, Max: 7.0000, Median: 4.0000, Average: 4.0000), a 'Color Map' dropdown (Blue=low, Red=High), checkboxes for 'Reverse Color Map Colors' and 'Use Discrete Color Map', a color gradient slider, and a 'Set Breakpoints from Metrics' button. At the bottom, there are settings for 'Attributes for All Objects in Style', including Layer (Default Layer), Stack Level (Middle), Immobile checkbox, Line Dashed checkbox, and a Set Font... button.

Attribute

Most of the attributes can have a **Default** value or a value that is dependent on the value of a selected field. When the **Use** box is checked for a particular attribute, the field selected in the **Field Lookup** is used to determine the value of the attribute based on either a Color Map or Value Map lookup. Otherwise, the default value will be used. Line Thickness, Total Area, Rotation Angle, and Rotation Rate attributes all use value map lookups. Line Color and Fill Color both use color map lookups. When determining the Field Lookup, either select the field from the drop-down or use the Find button to open the [Find field dialog](#) for easy searching.

When an attribute is selected, either by clicking on one of the options settings for that attribute or just clicking the background box for that attribute, the appropriate color map or value map for that attribute will be displayed in the right-hand panel.

Line Thickness

Thickness of the line making up the outside border of the display object.

Line Color


Color of the line making up the outside border of the display object.

Fill Color

Fill color of the display object. The Fill Color will be used only if the **Use Fill** box is checked.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite.

When using the Kite style, use of the two different Field Lookups can be selected independently and each has a separate color map. When not choosing to use one or both of the Field Lookups, that appropriate half of the Kite shape will use the default Fill Color choice. To switch between color maps for the two Field Lookup choices, either click on the field drop-down for the

appropriate half of the Kite or click the  button that will appear next to the field choices when the Kite style is selected.

Total Area

This determines the size of the display object. Check the option to **Use Auto Width of Objects** for the width of the objects to be determined automatically. If this option is not checked, the **Width** can be set manually. The height or size is always determined as a function of the width and can be set with the **Size to Width Ratio**. When using the Field Lookup table the total area of the object is determined based on the Characteristic Value returned. The dimensions of the object are then determined based on the total area and the Size to Width Ratio.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite. Even though two fields can be chosen when using the Kite style, only one value map is used for the Field Lookup. When using the Kite style and using the Field Lookup the total area of the object is based on the sum of the lookup values returned for both fields.

Rotation Angle

Static angle of rotation of a display object in degrees.

Rotation Rate

This value is given in Hz and indicates how many times per second an object should do a full rotation. This is used in conjunction with the [General Option for Animation Control](#). If animation control is not enabled, the rotation rate has no effect on how the objects are displayed.

Visibility

This attribute can prevent display objects from being shown if certain conditions are not met. Choosing the **All Visible** option will always show all of the display objects associated with this style. If choosing the option **Only visible if field is non-zero**, select a field from the drop-down box. Only those objects whose selected field is non-zero will be visible. If choosing the option **Only visible if filter is met**, select a filter from the drop-down box or click the Find button to search for an existing filter or create a new one. Only those objects that meet the selected filter will be visible. Note that the visibility attribute is only applied while in Run Mode. All objects will be visible in Edit Mode regardless of how this option is set.

Color Map

Color maps are used with Line Color and Fill Color attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the breakpoint values for the color map.

Color Map

Choose from the predefined color maps using the drop-down.

Reverse Color Map Colors

This checkbox reverses the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

Use Discrete Color Map

Check this box to make a discrete color map, that is without having smooth transitions between colors.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use Absolute Value

Check this box to use the absolute values of the field values when mapping the field values to colors.

Breakpoint Values

The breakpoint values determine how the field values will map to colors.

Set Breakpoints from Metrics

As different fields are chosen for attributes, the Field Set Metrics will be updated to reflect the field values. Click this button to update the breakpoint values based on a new set of Field Set Metrics.

Value Map

Value maps are used with Line Thickness, Total Area, Rotation Rate, and Rotation Angle attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the Field Values and corresponding Characteristic Values.

The attribute value for a particular display object is set to the Characteristic Value that corresponds to the field value of the corresponding data object. Characteristic values will be interpolated if the field value does not fall exactly on a value that has been defined in the table. To insert entries into the table, right-click on the table grid and choose Insert from the local menu. Choose Delete from the local menu to remove entries.

Line Thickness Value Lookup

Field Set Metrics

Min	1.0000	Median	4.0000
Max	7.0000	Average	4.0000

Field Value	Characteristic Value
1.000000	1
4.000000	2
7.000000	3

Attributes for All Objects In Style

These attributes apply to all objects independent of any specific field value.

Layer

Type in the name of an existing layer to assign the objects to an existing layer, or type in the name of a new layer to have that layer created and add the objects to that layer.

Stack Level

An object's stack level dictates what objects it will appear above and which objects it will appear below on a oneline display. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore, anything with a stack level of Middle, Background, or Base will appear underneath pie charts and circuit breakers on the oneline display. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the display.

Line Dashed

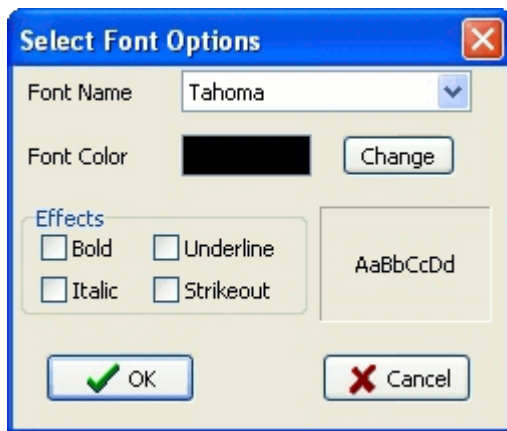
This determines the appearance of the line making up the outside border of the display object. This line can be solid or one of the dashed options.

Immobile

Check this box to keep the geographic display objects from being moved on the oneline diagram.

Set Font

Click this button to bring up the Select Font Options dialog. The changes to font are limited to Font Name, Font Color, and special Font Effects of Bold, Underline, Italic, and Strikeout. There is no option for setting the font size because this is automatically determined based on the size of the display object.



Geographic Data View Styles: General Display Options

The General Display Options associated with Geographic Data View Styles are applied to all geographic data view objects independent of any field value.

General Display Options | Fields and Attributes | Bus Records

Style

- ☒ Rectangle
- ☐ Rounded Rectangle
- ☐ Oval
- ☐ Triangle
- ☐ Kite

Object ID

Show Object ID by

- ☒ Name
- ☐ Number
- ☐ Name (Number)
- ☐ Number (Name)
- ☐ None

☐ Prefix ID with Object Type

Selection Options

☐ Can Be Selected

Line Thickness: 1

Line Color: [Green]

Animation Control

☐ Enable Animation

Interval (Milli-seconds): 100

Style

The style determines the shape of the geographic data view display object used to represent the selected data records.

Object ID

Each geographic data view display object can be identified by name, number, or a combination of the two. When a choice other than *None* is selected, the appropriate identification for that object will be displayed. When **Prefix ID with Object Type** is selected, the object type will be prefixed along with the main identifier. This option is useful when trying to distinguish between the different types of objects shown on a given display.

Selection Options

Geographic data view display objects can be formatted differently when they are selected as opposed to not selected. Check the option **Can Be Selected** to allow objects to be formatted based on the set options when they are selected. **Line Thickness** determines the thickness of the border line around the outside of the display object when it is selected. **Line Color** determines the color of the border line around the outside of the display object when it is selected.

Animation Control

Geographic data view display objects are capable of being animated if a [Rotation Rate](#) other than zero is specified with the [attributes](#). Check the **Enable Animation** box to animate the objects using this style. The **Interval** determines how often the display is refreshed during animation.

Geographic Data View Styles

Each geographic data view object has an associated Geographic Data View Style. The style determines how the object is going to be displayed and contains options for setting various attributes associated with the object. Each style refers to a particular type of object. Geographic data view objects cannot be individually formatted; they will always obey the formatting of their associated style.

The **Geographic Data View Options** dialog is used to create and modify styles that will determine the appearance of geographic data view objects. This dialog is part of the [Geographic Data View Customization](#) dialog when adding new geographic data view objects to a display, but it becomes a stand-alone dialog when modifying the styles for existing objects. To open as a stand-alone dialog, right-click on an existing geographic data view object and choose **View Geographic Data View Options** from the local menu.

Usually, groups of objects share the same geographic data view style. Keep in mind that changing the options in a style will change all of the objects that use this style.

Geographic Data View Options Dialog

The Geographic Data View Options dialog is made up of four main sections and tabs:

Object and Field Display Option Sets



The screenshot shows the 'Object and Field Display Option Sets' dialog box. It has a title bar with the same text. Inside, there are two input fields: 'Object Type' with the value 'Bus' and 'Style Name' with the value 'Bus_BusNum'. To the right of these fields are four buttons: 'Save', 'Save As', 'Rename', and 'Delete'. Below these buttons are two more buttons: 'Save to Auxiliary' and 'Load from Auxiliary'.

This section allows the saving, renaming, and deleting of Geographic Data View Styles. Only those styles for a particular **Object Type** will be displayed in the **Style Name** dropdown box. Style names must be unique within an Object Type. Geographic data view styles can be stored in the standard [auxiliary file](#) format. When saving or loading an auxiliary file from this dialog, all geographic data view styles will be stored or loaded regardless of the current object type. The default is to store the information in a data auxiliary file, .AUX, but these objects are also supported in display auxiliary files, .AXD.

[General Display Options](#)

[Fields and Attributes](#)

Object Records

The object records tab lists the data objects that are linked to the geographic data view style, through their associated geographic data view display objects, selected when the dialog is opened. The caption of this tab will be updated to reflect the class type of object that is represented. For example, if viewing bus objects, the caption will be *Bus Records* instead of *Object Records*. These are either the data objects selected before creating a new geographic data view or data objects represented by existing geographic data view objects that are using the selected style. The format of the display objects represented by these data objects will be changed based on any changes made to the options. To assign a group of geographic data view display objects to a new style, simply select a different style than the style in place when opening the dialog and click either the **OK** button on the **Geographic Data View Options** dialog (existing data object was selected) or the **Create Geo Data View Display and Close Dialog** button on the **Geographic Data View Customizations** dialog (new geographic data view objects are being created).

To change the style of a single geographic data view display object and assign it to a new style, use the Display Explorer. Within the Display Explorer, [case information displays](#) for geographic data view objects are available. The style is given in the field **Geographic Data View Style Name** (variable

name GEODATAVIEWSTYLENAME). To change the style, change the name listed in this field to a style that is available for the type of object being changed.

Custom Case Information Display

To open the Custom Case Information Displays, go to the [Case Information](#) ribbon tab, and choose **Custom Case Info** from the **Case Data** ribbon group

Custom Case Information Displays can be used to create a display very similar in appearance to a spreadsheet workbook with several worksheets. This display can show any information that can be shown on a [case information display](#), but also allows you to customize the layout of the information in any manner.

There are three buttons which allow you to **Rename** the present sheet, add a **New** sheet, or **Delete** the present sheet.

There are three different kinds of cells allowed on this display

- Blank Cell – contains nothing
- Plain Text Cell – contains a user-entered string with no link to any data in the model.
- Model Field Cell – contains a link to model field similar to inserting a [model field](#) on a oneline diagram

Each cell of the display will behave differently depending on the Custom Case Info Mode. There are four distinct modes that control the user interaction and operation of the custom case information display. These four modes and the effect they have on the display are described in topics

- [Define Fields/Strings](#)
- [Change Field Data](#)
- [Show Fields Primary](#)
- [Show Fields Secondary](#)

Define Fields/Strings

The "Define Fields/String" mode is the primary mode for use when setting up a new [Custom Case Information Display](#). The three kinds of cells behave as follows in this mode.

Blank Cell

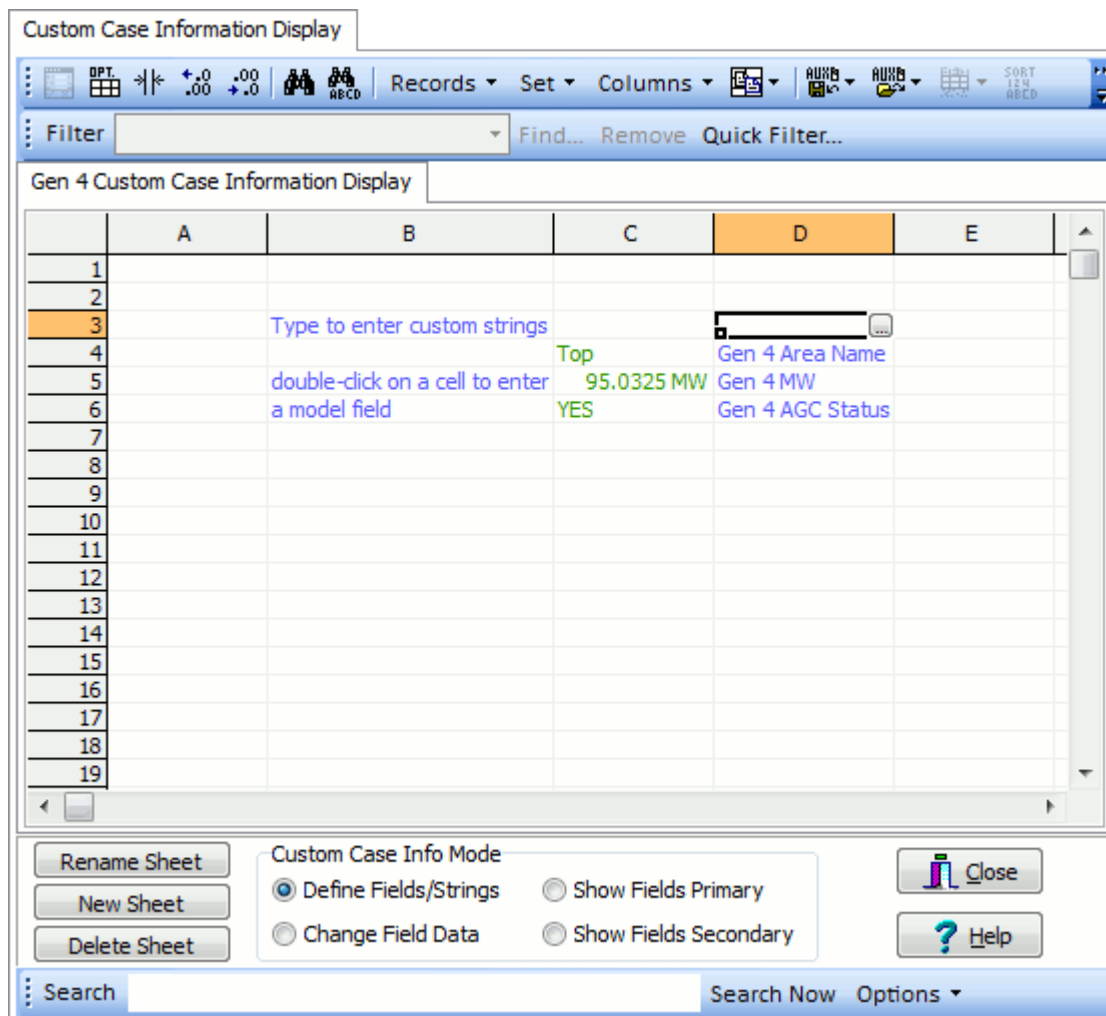
To convert to a Plain Text Cell, just type in the cell. To convert to a Model Field Cell, double-click on the cell to open up a dialog for defining the model field.

Plain Text Cell

These cells will appear in the Case Info Display Enterable Color (blue by default). To change them just type on the cell. When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell

Model Field Cell

Model Field cells will not be enterable. In order to edit these fields, you must double-click on the cell to open a dialog for defining the model field.



Change Field Data

The "Change Field Data" mode can be used to edit the data referred to by the model fields on a custom case information display. The three kinds of cells behave as follows in this mode.

Blank Cell

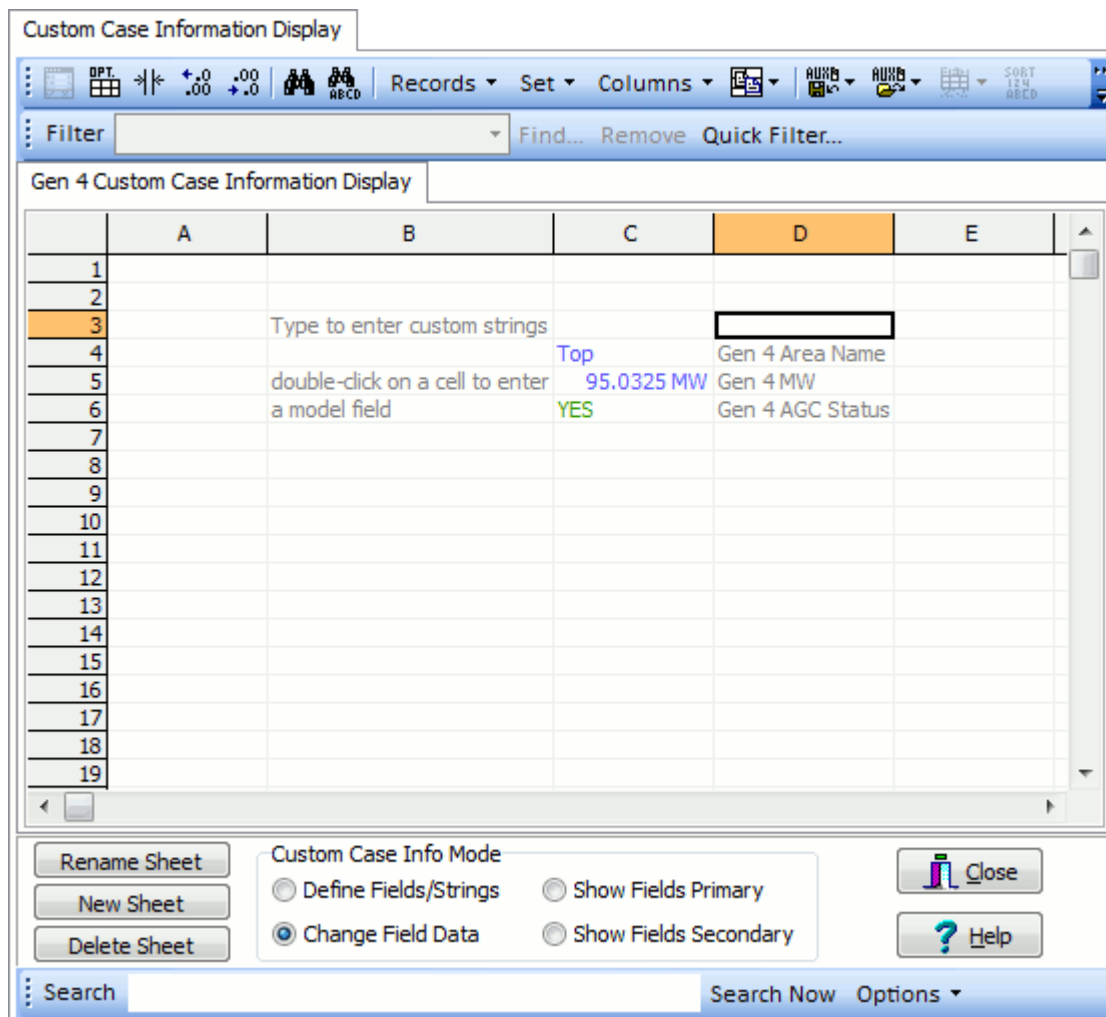
Blank cells may not be edited in any manner.

Plain Text Cell

Plain Text cells may not be edited in any manner. It will appear in a special color defined for the custom case information display which may be specified from the [local menu](#) of the workbook tabs. By default this color is dark gray.

Model Field Cell

Model Field cells will behave according to the field to which they refer. Enterable fields will be enterable, toggleable fields will toggleable, etc... When pasting in the sheet in this mode you will be modifying the model data directly. In the following picture there are three fields in cells C4, C5, and C6 that refer to read-only, enterable, and toggleable field respectively. The fields are labeled using the plain text cells in D4, D5, and D6.



Show Fields Primary

The "Show Fields Primary" mode is used for interacting with an external spreadsheet to create the model field links in custom case information displays. The three kinds of cells behave as follows in this mode.

Blank Cell

Blank cells behave the same as for the "[Define Fields/Strings](#)" mode.

Plain Text Cell

Plain Text cells behave the same as for the "[Define Fields/Strings](#)" mode.

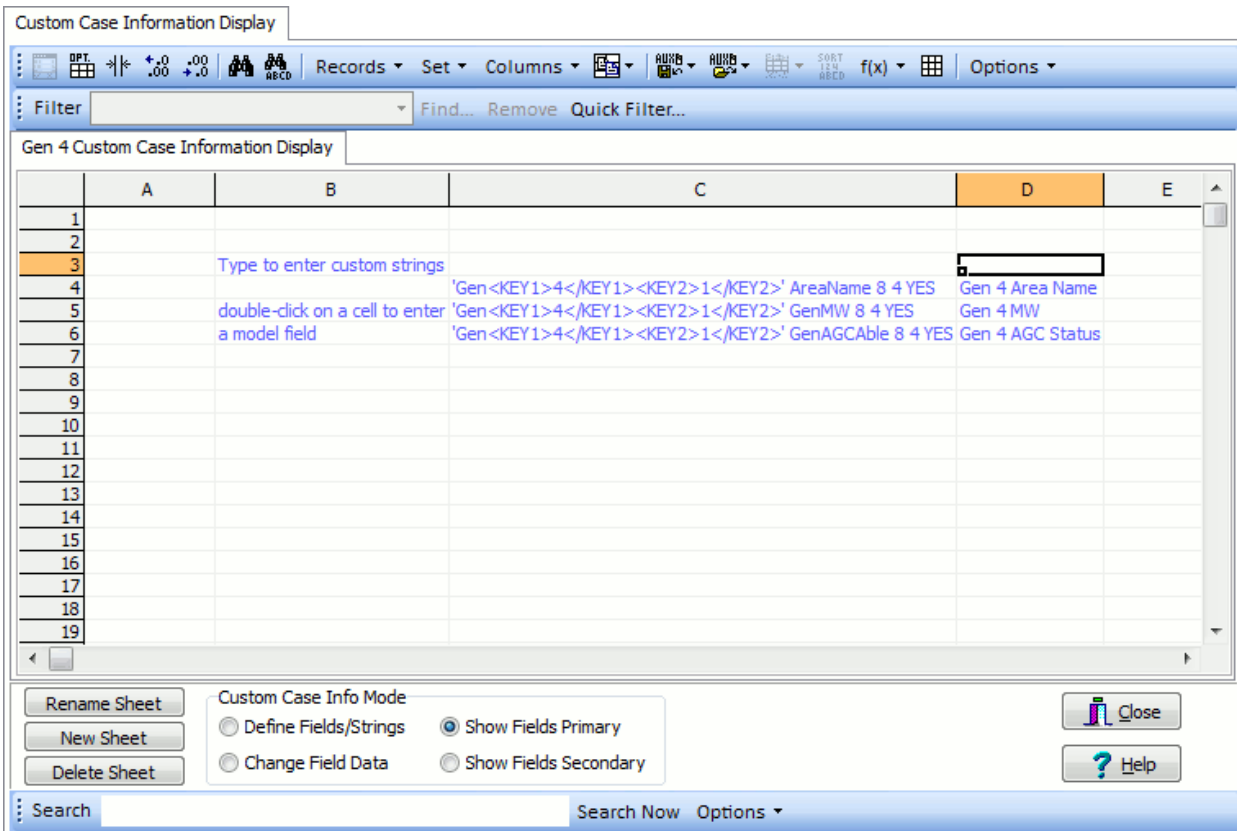
Model Field Cell

Model field cells will display a string which represents information about the model field link. The format for this string will be

'model field name' 'variable name' totaldigits decimalpoints IncludeUnits

The model field string will use the primary [key fields](#).

When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell.



Show Fields Secondary

This mode is identical to the [Show Fields](#) mode, except that it will show model fields string using the secondary [key fields](#).

Custom Case Information Display

Filter Find... Remove Quick Filter...

Gen 4 Custom Case Information Display

	B	C	D	E
1				
2				
3	Type to enter custom strings			
4		'Gen<KEY1>Four_138.00</KEY1><KEY2>1</KEY2>' AreaName 8 4 YES	Gen 4 Area Name	
5	double-click on a cell to enter	'Gen<KEY1>Four_138.00</KEY1><KEY2>1</KEY2>' GenMW 8 4 YES	Gen 4 MW	
6	a model field	'Gen<KEY1>Four_138.00</KEY1><KEY2>1</KEY2>' GenAGCable 8 4 YES	Gen 4 AGC Status	
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				
19				

Rename Sheet New Sheet Delete Sheet

Custom Case Info Mode

☐ Define Fields/Strings
 ☐ Show Fields Primary
 ☒ Show Fields Secondary
 ☐ Change Field Data

Close Help

Search Search Now Options

Custom Case Information Display Local Menu

By right-clicking on the tabs representing the various sheets of the [custom case information display](#), a local menu appears giving you the following options.

- Rename Sheet – this is only available if you click directly on a specific tab. Choose it to rename the sheet.
- Delete Sheet – this is only available if you click directly on a specific tab. Choose it to delete the sheet.
- New Sheet – Choose this to add a new sheet
- Save All Sheets... – Choose this save all the information on the sheets to an auxiliary file. For the fields one the display, all information will be saved to the auxiliary file using [Primary Key Fields](#) unless the Custom Case Info Mode is set to Show Fields Secondary. In this case, they will be saved using the secondary key fields.
- Custom Case Info Mode – Choose this to open a submenu that allows you to change the mode.
- Tabs Position – Choose this to open a submenu that allows you to change the location of the tabs relative to the sheets
- Plain Text Change Data Color – Choose this to change the color which is used to denote Plain Text cells when the Custom Case Info Mode is set to [Change Field Data](#). The default color is dark gray.

By right-clicking on one of the Custom Case Information Display sheets, you will open a local menu that has many options which are the same as other [Case Information Displays](#). There are a few extra options listed as follows

- Insert Row – Choose this to insert a new row at the location of the presently selected cell
- Delete Row – Choose this to delete rows that are part of the present selection
- Insert Column – Choose this to insert a new column at the location
- Delete Column – Choose this to delete columns that are part of the present selection
- Delete Cell – Choose this to delete the presently selected cells
- Custom Case Info Menu – Choose this to open the same menu available by right clicking on the tabs representing the various sheets

User-Defined Case Information Displays

While it is normally sufficient to customize the case information displays that come with the program, it is sometimes convenient to create different sets of columns, filtering, sorting, etc... for the same object type. One set of generator fields may be convenient for studying voltage regulation problems, while another set of generator fields may be convenient for looking at MW-related information. This ability is provided using the User-Defined Case Information Displays. This provides the ability to switch between different sets of columns, filters, sorting etc... without requiring repeated customizations.

Note: User-Defined Case Information Displays will not show any of the various Record-Specific Actions which often appear in the [Case Information Toolbar Records Menu](#). To access these special options, the normal case information display must be used instead.

The management of User-Defined Case Information Displays is all done from the [Model Explorer: Explore Pane](#).

The screenshot shows the 'Model Explorer: Generators' window. The 'Explore' pane on the left lists various object types, with 'User-Defined' selected. A context menu is open over the 'Explore' pane, showing options like 'Toolbar (Top)', 'Searchbar (Bottom)', 'Filterbar (Top)', 'Open Multiple Explorers', 'Save Recent (10)', 'Show Number of Objects', 'Insert User-Defined Case Info', and 'Remove User-Defined Case Info'. The 'Insert User-Defined Case Info' option is highlighted. The main pane displays a table of user-defined case information displays.

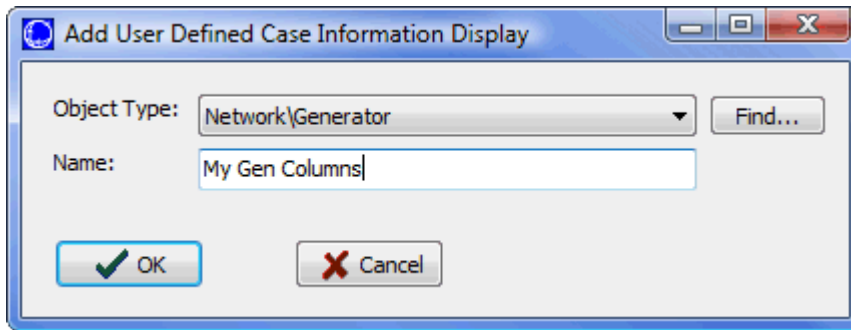
Right-Click on Explore Pane to insert a User-Defined Case Information Display

Number of Bus	Name of f	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor
1	46671 ALDER11	1	Closed	10.00	6.02	1.02899	NO	YES	0.00	26.00	0.00	13.00	None	26.00
2	46672 ALDER12	1	Closed	10.00	6.02	1.02899	NO	YES	0.00	26.00	0.00	13.00	None	26.00
3	47734 BHP 30	30	Open	0.00	0.00	1.01594	NO	YES	0.00	45.00	-15.00	35.00	None	45.00
4	47735 BHP 40	40	Open	0.00	0.00	1.01594	NO	YES	0.00	45.00	-15.00	35.00	None	45.00
17	40307 COWFALLS	1	Closed	140.00	106.46	1.07000	NO	YES	0.00	240.00	-82.00	123.00	None	240.00
18	46789 CUSHMN11	1	Closed	17.00	10.00	1.03913	NO	YES	0.00	35.00	-10.00	10.00	None	35.00
19	46790 CUSHMN12	2	Open	0.00	0.00	0.98261	NO	YES	0.00	23.00	0.00	10.00	None	23.00
20	46791 CUSHMN2	3	Open	0.00	0.00	0.98261	NO	YES	0.00	30.00	0.00	12.00	None	30.00
21	46791 CUSHMN2	2	Open	0.00	0.00	0.98261	NO	YES	0.00	30.00	0.00	12.00	None	30.00
22	46791 CUSHMN2	1	Closed	25.90	12.00	0.98261	NO	YES	0.00	30.00	0.00	12.00	None	30.00
23	46419 DIABLO31	1	Open	0.00	0.00	1.00000	NO	YES	0.00	83.00	-17.50	35.00	None	83.00
24	46420 DIABLO32	2	Open	0.00	0.00	1.00000	NO	YES	0.00	83.00	-17.50	35.00	None	83.00
25	42711 ELECTRON	2	Closed	1.98	0.00	1.00000	NO	NO	0.00	3.50	0.00	0.50	None	3.50
26	42711 ELECTRON	1	Closed	1.98	0.00	1.00000	NO	NO	0.00	3.50	0.00	0.50	None	3.50
27	42711 ELECTRON	3	Closed	1.98	0.00	1.00000	NO	NO	0.00	3.50	0.00	0.50	None	3.50
28	42711 ELECTRON	4	Closed	5.98	0.00	1.00000	NO	NO	0.00	9.40	0.00	1.00	None	9.40
29	42011 ENSERCH1	1	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
30	42012 ENSERCH2	2	Open	0.00	0.00	1.01700	NO	YES	0.00	38.30	-8.27	25.16	None	38.30
31													None	38.30
32													None	58.20
33													None	74.40
34													None	74.40
35													None	104.00
36													None	104.00
37													None	68.00

List of User-Defined Case Information Displays

Inserting User-Defined Case Information Displays

To define a User-Defined Case Information Display, right-click on the [Model Explorer: Explore Pane](#), and choose **Insert User-Defined Case Info**. On the dialog which appears click **Find...** to change the **Object Type** and then enter a **Name** for the new display.



Viewing User-Defined Case Information Displays

A list of User-Defined Case Information Displays are shown in the folder **User-Defined** at the bottom of the [Model Explorer: Explore Pane](#).

Removing User-Defined Case Information Displays

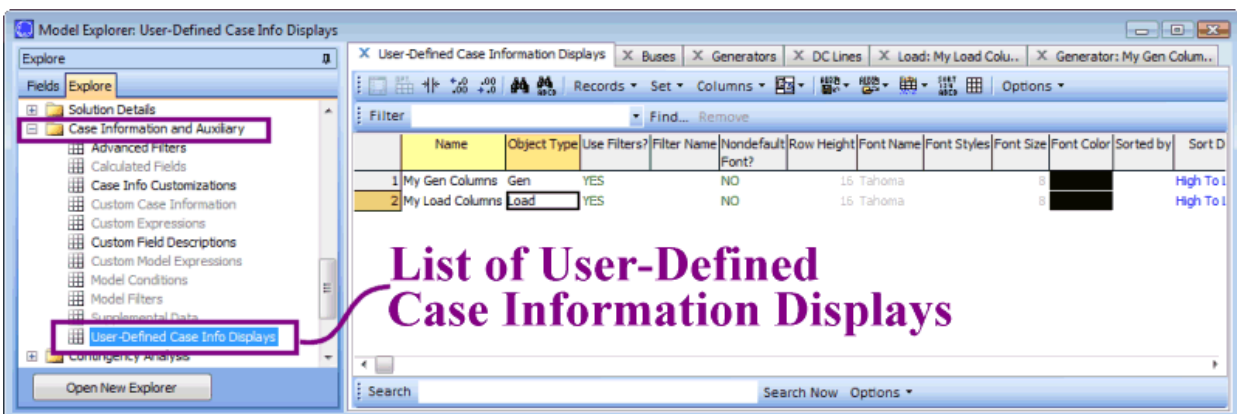
To remove a User-Defined Case Information Display, right-click on its entry under the **User-Defined** folder on the Explore Pane. Then choose **Remove User-Defined Case Info**.

Customizing User-Defined Case Information Displays

The customization of a User-Defined Case Information Display is exactly the same as all other case information displays. See the [Configuring Case Information Displays](#) for more information.

Saving Settings for User-Defined Case Information Displays

Settings for the User-Defined Case Information Displays can be saved in the [Auxiliary File Format](#). To save these, in the Explore Pane, go to the Case Information and Auxiliary/User-Defined Case Info Displays. This will open a case information display showing the user-defined case information displays. Then save to an auxiliary file using the [Case Information Toolbar: Save Auxiliary Files Menu](#).



Case Description

The Case Description Dialog is broken into two sub-tabs

Case Description

The Case Description Tab allows you to enter a text description of a case. The portion of the description that is saved with the case varies with case type:

PowerWorld Binary (*.pwb) an unlimited number of lines are allowed

PTI Raw Data format (*.raw) Two lines of text

GE EPC Data format (*.epc) The case title and all the pre-title comments are read in as the case description

PowerWorld Case (*.pwc) No case description supported

IEEE Common Format (*.cf) No case description supported

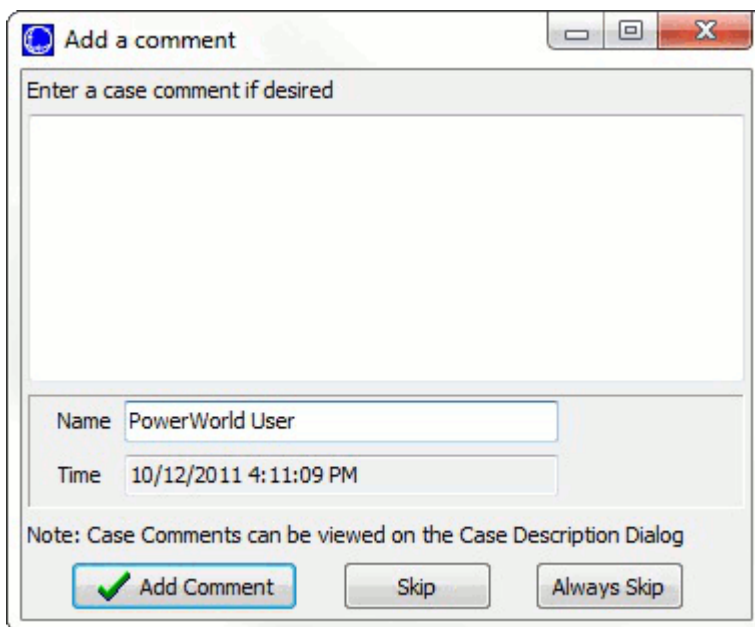
In PowerWorld Viewer, these descriptions are read-only.

Case Comments

The Case Comments Tab allows you to read comments previously added to a particular case. It shows a Case Information type of table with the Date, Time, User and Comment. This info is useful to track particular changes to a case or any information that need it to be commented after saving a case.

Clear Comments will clear all of the comments associated with the respective case.

Checking the **Prompt for comment when saving case** checkbox will force saving the case with comments by popping a dialog every time a case is saved. The portion of the description that is saved with the case varies with case type. The Add a comment dialog is shown below:



Add Comment will add the comment to the particular case.

Skip will skip the adding of the comment for the case at this particular time is saved.

Always Skip will disable this dialog from appearing in future savings. To re-enable the dialog prompt, the **Allow to prompt for comment when saving case** option can be set with [Environment Options](#).

The Comment can be written in the white box and the Name can be modified also.

The dialog is slightly modified if [forcing the saving of comments when saving a case](#). A **Prompt for comment when saving case** checkbox will be available to set this option instead of doing that with the Environment Options.

To display the Case Description Dialog to the [Case Information](#) ribbon tab, and choose **Case Description** from the **Case Data** ribbon group.

Case Summary

The Case Summary Display provides a summary of the current case. Note that there are no enterable fields on the display. To display the Case Summary, go to the [Case Information](#) ribbon tab, and choose **Case Summary** from the **Case Data** ribbon group. The fields shown on this display include:

Number of Devices in Case shows the number of each of the following device types:

Buses

Total number of buses in the case. Use the [Bus Display](#) to see a listing of these buses.

Generators

Total number of generators in the case. Use the [Generator Display](#) to see a listing of these generators.

Loads

Total number of loads in the case. Use the [Load Display](#) to see a listing of these loads.

Switched Shunts

Total number of switched shunts in the case. Use the [Switched Shunt Display](#) to see a listing of these switched shunts.

Trans. Lines (AC)

Total number of transmission lines in the case. Use the [Line/Transformer Display](#) to see a listing of these lines.

LTCs (Control Volt)

Total number of Load Tap Changing transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Phase Shifters

Total number of Phase Shifting transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Mvar Controlling

Total number of Mvar Controlling transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

Series Capacitors

Total number of Series Capacitors in the case. These devices are treated the same as any other transmission line, and therefore can be found listed as a transmission element in the [Line/Transformer Display](#).

2 Term. DC Lines

Total number of two-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

N-Term. DC Lines

Total number of multi-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

Areas

Total number of areas in case. Use the [Area Display](#) to see a listing of these areas.

Zones

Total number of zones in the case. Use the [Zone Display](#) to see a listing of these zones.

Islands

Total number of islands in the case. An island is a group of buses that are interconnected through ac transmission lines or transformers but are isolated from the rest of the system. Each island must have a slack bus. In Simulator, use the Power Flow Solution tab of the [Simulator Options display](#) to specify whether multiple islands are allowed.

Interfaces

Total number of interfaces in the case. An interface is a grouping of tie line objects between area objects. In Simulator, use the [Interfaces Display](#) to open the [Interface Dialog](#) to define and modify interface objects.

Injection Groups

Total number of injection groups in the case. An injection group is a collection of loads and generators (objects that inject power into a network). In Simulator, use [Injection Groups Display](#) to open the [Injection Groups Dialog](#) to define and modify injection groups.

Case Totals

Summarizes the total load, generation, shunt compensation, and losses for the case. Positive shunt compensation denotes shunt load, whereas negative shunt compensation indicates a shunt injection (such as shunt capacitance). The case totals fields are valid only when the current case is solved.

Generator Spinning Reserves

The total difference between the present total generator output versus the total maximum possible output of all in-service generation.

Slack Buses:

The slack bus or buses are listed showing the bus name, number, and area name and number. One slack bus is required for each island.

Case Pathname

Full file name of the current case.

Power Flow List

The Power Flow List shows detailed information about the system's power flows in a more traditional text-based form. This information is intended for users who would like detailed flow information about the power flow. Including the per unit voltage at the bus, the bus' load and generation, and flows on all lines and transformers emanating from the bus. The content of this display (i.e. which buses are included in the list) is governed by the [area/zone/owner filters](#).

To show this display go to the [Case Information](#) ribbon tab, and choose **Power Flow List** from the **Case Data** ribbon group.

To view flows at just a few select buses you may want to use the [Quick Power Flow List](#) instead. For large systems with no area/zone filtering set, it may take Simulator a long time to generate the complete Power Flow List. Note also that this display can show a maximum of 32,767 lines of text. If this limit is exceeded, Simulator will generate a resource error. Either use the [area/zone/owner filters](#) to limit the number of devices shown on this display, or use the Quick Power Flow List to focus on a few selected buses of interest.

The Power Flow List allows you to navigate through the system's buses rather easily. You can also use the display to show the flows for a bus' neighbor by double-clicking on the line that reads "TO nnnnn...", where nnnnn is the number of the bus you would like to see. The display is then positioned at this bus. If the bus is in an area and/or zone whose area/zone filter is not set, the area/zone filter is set automatically. In this way, you can inspect the system bus by bus.

The Power Flow List also has its own local menu, which can be viewed by clicking the right mouse button on the display. Select **Change Font** to modify the style and size of the display's font. Select **Refresh** to ensure that the display's contents concur with the current system state. To skip to particular bus in the list, click **Find Bus**, which will open the [Find Bus Dialog](#). To display the information dialog for the currently selected branch, bus, generator, load, or switched shunt, select **Show Dialog**. To print the display, choose the **Print** local menu option. Choosing **Copy** enables you to copy the display into the Windows clipboard, from where the information can be pasted into another application. Finally, select **Close** to close the display.

When printing the display you can either send the results directly to the printer or save them to a text file. To save the results in a text file on the Print Dialog select the **Save to File** option shown in the lower left corner of the dialog.

For each bus, the following items are shown:

Bus

Shows the bus' number, name, and nominal voltage in kV. The next four fields are the MW, MVar, MVA and percentage headers for subsequent rows. The next fields specify the per unit voltage magnitude, voltage angle in degrees, the bus' area number and the bus' area name.

Generator

For each generator at the bus, the Power Flow List shows the generator's ID (immediately after the keyword GENERATOR) and the power output of the generator in MW. Following this is generator's reactive power output in Mvar. A single character is shown immediately after the Mvar field. An 'R' indicates that the generator is regulating the bus voltage, 'H' indicates that the generator is at its high reactive power limit, 'L' indicates that the generator is at its low reactive power limit, and a blank suggests that the generator is set off of AVR. The last field in the GENERATOR item is the MVA output of the generator. If no generators are connected to the bus, this item will be absent from the display.

Load

Shows the total power consumed by each load at the bus. If no loads are present at the bus, this item will be absent from the display.

Shunt

Shows the total power for the fixed shunts at the bus. Positive shunt values denote shunt load, while negative shunt quantities indicate injection. If no shunts are connected to the bus, this item will be absent from the display.

Switched Shunt

Shows the total power for the switched shunts at the bus. If no switched shunts are located at the bus, this item will be absent from the display.

Lines and Transformers

For each line or transformer coming into the bus, the Power Flow List shows the line's flow and percentage loading. For transformers, the off-nominal tap ratio and phase shift angle in degrees are also shown. Immediately to the right of the off-nominal tap ratio is a two-character designation indicating the tapped side of the transformer: 'TA' indicates that the bus is on the tapped side, while 'NT' identifies the bus as residing on the side without the tap. You can left-click on this field to immediately reposition the bus to the other end of the line or transformer.

Quick Power Flow List

The Quick Power Flow List provides a convenient means of viewing a listing of the flows at individual buses in the system. The format and control of the Quick Power Flow List is generally the same as that of the [Power Flow List](#), except that the Quick Power Flow List displays results for just the desired bus or [range of buses](#).

You can access this Quick Power Flow List in a number of different ways:

- Go to the [Case Information](#) ribbon tab, and choose **Quick Power Flow List** from the **Case Data** ribbon group
- From most of the Case Information Displays, right-click to invoke the display's local menu, and select *Quick Power Flow List*.
- From the [Bus Information Dialog](#) click on the *View All Flows at Bus* button.
- From the oneline diagram, right-click on the [bus symbol](#) to display the bus' local menu, and select *Quick Power Flow List*.

This display is automatically created if it is not already shown. Information on subsequent buses appears at the bottom of the display.

As with the Power Flow List, you can navigate through the system bus-by-bus by double-clicking on the lines that begin with "TO nnnnn ...," where *nnnnn* is the number of the bus you would like to investigate. Information for that bus will appear at the bottom of the display.

Like the Power Flow List, the Quick Power Flow List has a local menu that is accessed by right-clicking on the display. Among the things you can do from the local menu is to display the bus, branch, generator, load, or shunt corresponding to the currently selected record by selecting **Display Object Dialog**. You can also navigate through the system bus-by-bus just as you can do through double-clicking by choosing **Goto Line Bus** from the local menu.

Outages

The outages display, available by choosing **Solution Details > Outages** in the [Model Explorer](#), presents a tabular listing of devices that are currently out-of-service in the load flow model. The display contains pages that resemble case information displays for [branches](#), [generators](#), [loads](#), [switched shunts](#), [buses](#), and [Multi-section lines](#). All devices listed in the pages on this display are the out-of-service elements.

Area Display

The Area Display houses data about each area in the case. The Area Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

To show the area records display, select **Aggregations > Areas** from the [Model Explorer](#) or Aggregation > Areas from the Case Information ribbon group under the [Case Information](#) ribbon tab.

By default, the area records display contains the following fields:

Number, Name

Area's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the area names will be truncated at their maximum supported length.

AGC Status

Area's automatic generation control status. This field indicates whether or not the area's generation is changing automatically to control the area interchange. See [Area Control](#) for more details. You can toggle the value of the area's AGC Status (except in Viewer) by left clicking on the entry. Valid entries in this field include:

Off AGC Area is off AGC. Generation must be adjusted manually to meet changes in load and losses. If it is not, the system slack will be forced to pick up the balance.

Part AGC Area is on AGC, with generation dispatch controlled by its units' [participation factors](#).

ED Area is on economic dispatch control so that generation is dispatched in order of least cost.

OPF Area is on OPF control (only used with Simulator OPF). This option is only used with Simulator OPF. When the case is solved using the OPF the area controls are changed by the OPF to maintain area power balance. During non-OPF solutions this option is equivalent to Off AGC.

Area Slack Area is on Area Slack control. This option is only available for an area if you have already specified an area slack bus number in the area's [information dialog](#).

Gen MW

Total real power generation in the area in MW.

Load MW

Total real power load in the area in MW.

Tot Sched MW

The net of the base and scheduled transactions between the area and all other areas, with exporting power indicated as a positive value. The interchange for an area is set on the [Area Information Dialog](#); the MW transactions for an area can be viewed using the specified [MW Transactions display](#).

Int MW

The actual interchange between this area and all other areas, with exporting power positive. If an area is on AGC control, its actual interchange should match its scheduled interchange.

ACE MW

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

Lambda

The area's marginal cost. This marginal cost is relevant only when an Economic Dispatch (not OPF) solution has been run. Theoretically, if losses are ignored, an area operates most economically if all generators operate at the same incremental cost. This common incremental cost is the area's *lambda*, or marginal cost.

The local menu of the area records display has an additional option labeled *All Area Gen IC Curves*. Select this activity to generate a plot showing the incremental cost curves for all units located in a particular area.

Loss (MW)

Total real power losses for the area.

Losses for tie-lines between areas are assigned to the area in which the terminal bus that is NOT the metered bus is contained. The **Metered End** field for a branch determines which of the terminals is metered.

Auto Shunts

Determines whether switched shunts for the area are available for automatic control. You can use this field to disable all the switched shunts in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set the auto shunts property for all switched shunts. Note that a switched shunt is available for automatic control only if it meets three conditions: 1) its control mode property is set to *Automatic* (see [Switched Shunt Information Dialog](#)); 2) its associated area's *Auto Shunts* property is set to *Yes*; and 3) the *Disable Switched Shunt Control* option on the Power Flow Solution tab of the [Simulator Options Dialog](#) must not be checked.

Auto XF

Determines whether tapped transformers for the area are available for automatic control. You can use this field to disable all the transformers in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set all entries in this column. Note that three conditions must be met for a transformer to be used for automatic control: 1) its *Auto* field must be set to *Yes* (see [Transformer Modeling](#) for details); 2) its associated area's *Auto XF* property is set to *Yes*; and 3) the *Disable Transformer Control* option on the Power Flow Solution tab of the [Simulator Options dialog](#) must not be checked.

Unspec. MW Inter.

Shows the total amount of interchange for the area that is listed as Unspecified. This means that some of the interchange is known in magnitude and direction (import or export) but not which other area(s) is involved in the interchange. The sum of all area's unspecified interchange in the case must equal 0 for a balanced system.

Zone Display

The Zone Display provides information about all the zones in the case. Similar to the [Area Display](#), the Zone Display provides a means of dividing up a power system. System results can then be summarized by zones using this display. Buses can be assigned to zones independent of their area assignments. Thus a single area could contain multiple zones, or a single zone could span multiple areas. The zone number for each bus is shown on the [Bus Dialog](#). In the Edit Mode, groups of buses can be easily moved from one zone to another using the [Zone Dialog](#).

The Zone Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated zones. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about a representative bus in the zone. You can also sort the zone records by clicking on the heading of the field by which you want to sort.

To show this display, click on **Aggregations > Zones** in the [model explorer](#).

The display contains the following fields by default:

Zone Number, Zone Name

Zone's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the zone names will be truncated at their maximum supported length.

Load MW, Load MVR

Total real and reactive power load in the zone.

Gen (MW), Gen (Mvar)

Total real and reactive power generation in the zone.

Loss MW, Loss MVR

Total real and reactive power loss in the zone. Losses are computed by summing the losses of the individual transmission lines and transformers in the zone. Because of shunt charging, these devices can also generate reactive power. Therefore, reactive power losses may actually be negative.

Losses for tie-lines between zones are assigned to the zone in which the terminal bus that is NOT the metered bus is contained. The **Metered End** field for a branch determines which of the terminals is metered.

Int MW, Int Mvar

Net interchange of real and reactive power with all other zones. Exported power is assumed to be positive.

Load Mult MW

Shows the current load MW multiplier for the zone. All load in the area is scaled by this multiplier.

Tie Lines between Areas Display

The Tie Lines between Areas Display identifies all area tie lines in the case. Tie lines are not a separate data record that can be created in Simulator, but instead provide a look at all the devices that connect two areas together in the power system model. Tie lines are important because the [Area Control](#) algorithms used by Simulator often enforce a constraint which requires that the sum of the flow on the tie lines must equal the net export scheduled by the area (This is often called the ACE equation).

There are several types of devices that can represent tie lines. These include the following:

- Transmission branch (Branch) : a transmission line or transformer that connects two buses that belong to two different areas is a tie line.
- Two-terminal DC transmission lines (DCLine) : a DC transmission line that connects two buses that belong to two different areas is a tie line.
- Multi-terminal DC transmission line (MTDCRecord) : For every multi-terminal DC transmission line that has converters connected to more than one area is a tie line.
- Load (Load): A load can be assigned to a different area than its terminal bus. This is not typical but is sometimes done to model loads at lower voltage level where more than one entity has load that connects to the same power system bus. In such a situation this load represents a tie line.
- Generator (Gen): Similarly to the load, a generator can be assigned to a different area than its terminal bus. Again this is considered a tie line.
- Switched Shunt (Shunt): Similarly to the load, a switched shunt can be assigned to a different area than its terminal bus. Again this is considered a tie line.

Note: The easiest way to see which loads, generators, or switched shunts are assigned in this way is to use this Tie Lines between Areas Display. You can also, however, add separate columns using the display/column options on the Load, Generator or Switched Shunt Case Information Display. For instance, for loads there are columns for Area Num of *Bus* and Area Num of *Load*. Normally these are equal, but they will not be for tie line loads.

Note: Be careful when working with cases opened as RAW files in which there are loads that represent tielines. See PTI RAW Transactions due to tie-line load in [Simulator Options: File Management](#) for more information.

The Tie Lines between Areas Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated device type. You can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the tie line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Tie Lines Display according to the constraints of the [area/zone/owner filters](#).

To call up the Tie Lines between Areas Display, select **Aggregation > Tie Lines between Areas** from the [Model Explorer](#).

The Tie Lines between Areas Display contains the following fields by default:

Tie Type

Indicates type of device that the tie line represents: Branch, Load, Gen, Shunt, MTDCRecord, or DCLine.

Near Area Name, Near Number, Near Name

The Bus Number, Name and Area Name of the bus to which the device is attached. For transmission lines this is the near end of the device.

Far Area Name, Far Number, Far Name

The Bus Number, Name and Area Name of the other end of the device.

Ckt

The circuit ID for transmission lines or DC Lines, the device ID for generators, loads, and switched shunts.

Meter MW and Mvar

The MW and Mvar flow from the point represented by Near Number/Name towards the Far Number/Name. For transmission lines and DC lines the metered point on the line is determined by the Metered End which is stored with the device. For instance on a transmission line, there is a check box for From End Metered which is available on the [Branch Options \(Edit Mode\)](#) dialog. Changing this setting with the device will affect the Meter values slightly.

Status

This is the Open/Closed status of the tie line device.

Lim MVA

For tie lines that represent transmission lines this is the limit (rating) of the device.

MW and Mvar Loss

For tie lines that represent transmission lines this is the MW and Mvar loss across the device.

Tie Lines between Zones Display

The Tie Lines between Zones Display is essentially identical to the [Tie Lines between Areas Display](#). The only difference is that it displays devices that connect two zones together instead of areas. See the [Tie Lines between Areas Display](#) for more details.

Super Area Display

The Super Area Display identifies any super areas that have been defined for the case. Super areas are groups of areas whose generators are dispatched as a coordinated group. Super areas can thus be useful for modeling the operation of independent system operators or power pools.

Super areas cannot be inserted into a case from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Instead, a super area can be defined when modifying or creating an area simply by typing the name of a new super area in the *Super Area* dropdown box on the [Area Information Dialog](#).

The Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated super areas. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about representative bus in the super area. You can also sort the super area records by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregation > Super Areas** in the [Model Explorer](#).

The display contains the following fields by default:

Super Area

The name of the super area.

AGC Status

The Super Area may operate without automatic generation control (AGC Status = *Off AGC*), with participation factor control (AGC Status = *Part. AGC*), or according to an economic dispatch (AGC Status = *ED*). This is a toggleable field.

Use Area PF

Indicates whether to use the areas participation factors when the super area is to operate with participation factor control (Part. AGC).

Num Areas

Indicates the number of areas defined as being part of the super area. Areas are added to super areas using the Super Area dropdown box on the [Area Information Dialog](#).

Gen MW

Total MW injection from all the generators in the super area.

Load MW

Total MW load demanded in the super area.

Tot Sched MW

Total scheduled MW interchange with other areas or super areas.

ACE MW

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

Lambda

The super area's marginal cost.

Loss MW

Indicates the real power losses incurred within the super area.

ED Use PF

Indicates whether the power flow engine will calculate loss penalty factors in computing the economic dispatch solution for the super area. If loss penalty factors are not calculated, then the economic dispatch is calculated assuming that the super area is lossless. Otherwise, the economic dispatch solution incorporates losses. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the super area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

Bus Display

The Bus Display presents data describing each bus in the case. The Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated bus. You can also sort the bus records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Bus Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the case information toolbar allows you to add new buses to or remove existing buses from the system.

To show the bus records display, select **Network > Buses** from the [Model Explorer](#) or from the Case Information ribbon group under the [Case Information](#) ribbon tab.

By default, the bus records display contains the following fields:

Number and Name

Bus number between 1 and 2,147,483,647 (equals 2^{31} minus 1), and its alphanumeric identifier.

Area Name

Alphanumeric identifier of the bus' area.

Nom kV

The nominal base voltage of the bus in kV.

PU Volt

Bus' per unit voltage magnitude.

Volt (kV)

Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

Angle (Deg)

Bus' voltage angle in degrees.

Load MW, Load Mvar

Total real and reactive load at the bus. If no loads are located at the bus, these fields are blank.

Gen MW, Gen Mvar

Total real and reactive generation at the bus. If no generators are located at the bus, these fields are blank.

Switched Shunt Mvar

Total switched shunt device reactive power injection at the bus.

Act G Shunt MW, Act B Shunt Mvar

Total real and reactive fixed bus shunt injections.

Area Num

The area number in which the bus is located.

Zone Num

The zone number in which the bus is located.

Bus Mismatches Display

The Bus Mismatches Display lists the real and reactive mismatches at each bus. The bus mismatches are defined as the difference between the power entering the bus and the power leaving the bus. A power flow case is considered solved when all the bus mismatches are below the convergence tolerance specified on the Power Flow Solution page of the [Simulator Options](#) dialog.

Most of the time you will not need to be concerned about the bus mismatches. If the power flow solves, the mismatches are guaranteed to be below the desired tolerance. However, advanced users will find this display useful in determining the cause when a power flow diverges.

The Bus Mismatch Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated bus. You can find a specific bus mismatch using the name or number of the bus, and you can learn more about the bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the bus mismatch information by clicking on the heading of the field by which you want to sort.

To show this display select **Solution Details > Mismatches** from the [Model Explorer](#). You can view bus mismatches only when the application is in Run Mode.

Also note that there is a special feature in the Records menu of the case information toolbar for the mismatch display. This option is **Zero-Out Mismatches**. Choosing this options will cause Simulator to insert fictitious B and G Bus Shunt values to force the mismatch at every bus to zero.

The Bus Mismatch Display contains the following fields by default:

Number, Name, Area Name

The number and name of the bus and the name of the area in which it is located.

MW Mismatch, MVR Mismatch, MVA Mismatch

The real and reactive mismatches at each bus, and the total complex power mismatch.

Remotely Regulated Bus Display

The Remotely Regulated Bus Display provides information about all buses that are remotely regulated by one or more generators, transformers, or switched shunts. The bus that a generator regulates is specified in the Edit Mode using the [Generator Dialog](#). Whenever a generator is regulating a bus that is not its terminal, it is considered to be remotely regulating that bus. The bus that is remotely regulated, along with the regulating generators, will appear on this display. The same logic is true for transformers and switched shunts which may regulate remote buses also.

The Remotely Regulated Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view [the information dialog](#) of its associated buses. You can find a specific remotely regulated bus, and you can learn more information about a particular remotely regulated bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the remotely regulated bus information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Remotely Regulated Bus Display, open the [Model Explorer](#) and select **Remotely Regulated Buses** under the Solution Details folder.

This Remotely Regulated Bus Display contains the following fields by default:

Number, Name, Area Name

Number, name, and area name for the bus that is being remotely regulated.

PU Volt

Per unit voltage magnitude for the bus.

Set Volt

Setpoint voltage for the bus. When a bus is being remotely regulated by a set of generators, the generators vary their reactive power output to maintain the voltage at the bus at the setpoint value. You can enter a new value for this field. Changing the setpoint voltage here changes the setpoint voltage for all the generators that are remotely regulating this bus.

Volt Diff

Per unit difference between the actual voltage magnitude and the set point voltage magnitude.

AVR

Combined automatic voltage regulation (AVR) status for all the generators remotely regulating this bus. If AVR is *No*, no generators regulate voltage; if AVR is *Yes*, all the available generators are regulating; if AVR is *Mixed*, some generators regulate voltage and some do not. Regulation of individual generators can be specified using the [Generator Display](#). You can toggle this field between "Yes" and "No" by clicking on it.

Total Mvar

Total of the reactive power being supplied by all the generators remotely regulating the bus.

MVR Min, MVR Max

Total of the minimum and maximum reactive power limits for all the generators remotely regulating the bus.

Remote Regs (gen)

Provides a comma-separated list of all the generators which regulate this bus.

Rem Regs (XFMR)

Provides a comma-separated list of all the transformers which regulate this bus.

Rem Regs (SS)

Provides a comma-separated list of all the switched shunts which regulate this bus.

Substation Records Display

The Substation Records Display presents data describing each substation in the case. The Substation Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated substation. You can also sort the substation records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Substation Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the local menu allows you to add new substations to or remove existing substations from the system. Note: a substation is considered in an area/zone if any single bus in the substation is in the area/zone.

To show the substation records display, select **Aggregation > Substations** in the [Model Explorer](#).

By default, the substation records display contains the following fields:

Sub Num

An integer identifier for the substation.

Sub Name, Sub ID

Two alphanumeric identifiers for the substation.

Area Name, Zone Name

The names of the area and zone of the buses in the substation. If some of the buses in the substation are in different areas or zones, then this is the most common area or zone.

of Buses

The number of buses inside the substation.

Nom kV

Nominal kV of the highest voltage bus(es) in the substation.

Gen MW, Gen MVR

Total real and reactive generation at the substation. If no generators are located at the substation, these fields are left blank.

Load MW, Load MVR

Total real and reactive load at the substation. If no loads are located at the substation, these fields are left blank.

Shunt MW, Shunt MVR

Total real and reactive shunt values at the substation. If no shunts are located at the substation, these fields are left blank.

Generator Display

The Generator Display presents data describing each generator in the case. The Generator Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Records Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator records display, select **Network > Generators** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for the bus.

ID

Single character ID used to distinguish multiple generators at the same bus. This default value for this field is '1'.

Status

Displays the Open / Closed status of the generator. This field is a toggleable field.

Gen MW, Gen Mvar

The real and reactive power output of the generator. If the generator is on AVR control, the reactive power is set automatically.

Set Volt

Per unit setpoint voltage for the generator. When a generator is on AVR control, the reactive power output of the generator is varied automatically in order to maintain the regulated bus voltage at this value. The regulated bus is usually, but not always, the generator's terminal bus. Use the [Generator Dialog](#) to see the regulated bus number.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

AVR

Designates whether the generator will vary its reactive power output to maintain a constant terminal voltage. If the *AVR* property is set to *Yes*, the generator is on automatic voltage regulation (AVR) control. When a generator is on AVR control, its reactive power output is varied automatically to keep the regulated bus voltage at the **Set Volt** value. AVR is limited by the generator's reactive power limits. You can click on this field to toggle its value (except in Viewer).

Min MW, Max MW

Minimum and maximum allowable real power output of the generator.

Min Mvar, Max Mvar

Minimum and maximum allowable reactive power output of the generator.

Cost Model

The type of cost model the generator is currently set to use. Cost models are necessary for performing economic analysis, such as Economic Dispatch or [Optimal Power Flow](#).

Part. Factor

Generator's participation factor. Participation factors are used to determine how AGC generators participate in area control when their area is on participation factor control. Please see [Area Control](#) for more details.

Generator Economic Curves

Four characteristic curves describe the efficiency and resulting costs associated with operating a particular generating unit. These four curves plot

- Fuel Cost
- Heat Rate
- Input-Output
- Incremental Cost

Simulator can display plots of all these curves. To display a particular plot for a generator, right-click on the generator in Run Mode to display its local menu, and then select the plot you wish to see. The plot will be presented in its own window. The windows for all plots exhibit identical characteristics. For example, the current operating point is identified by a red filled circle. Right clicking on an open area of the window displays the plot's local menu which allows you to print the plot, save it to a file, or copy it to the clipboard for use in other programs. To adjust the length and number of intervals shown on an axis, right-click on the *axis* (not on the *numbers*) then specify the min and max display values and number of intervals. To close a plot window, simply click the X button in its top right corner.

The Run Mode generator local menu also provides access to a fifth type of plot curve - the "All Area Gen IC Curves" plot. This plot simply shows the incremental cost curves and present operating points of all generators in the same area as the generator on which you clicked.

Fuel Cost Curve

The fuel cost curve specifies the cost of fuel used per hour by the generating unit as a function of the unit's MW output. This is a monotonically increasing convex function.

Heat-rate Curve

The heat rate curve plots the heat energy required per MWH of generated electrical output for the generator as a function of the generator's MW output. Thus, the heat rate curve indicates the efficiency of the unit over its operating range. Generally, units are least efficient at the minimum and maximum portions of their MW output capability and most efficient somewhere in the middle of their operating range. The vertical axis is plotted in MBtu/MWH and the horizontal axis is plotted in MW. You may interpret the heat rate for a generator producing X MW as follows: the heat rate indicates the amount of heat input energy per MWH of generation required to produce X MW of power. The lower this number, the less input energy is required to produce each MWH of electricity.

Input-Output Curve

The input-output curve is derived simply from the heat-rate curve by multiplying it by the MW output of the unit. This yields a curve showing the amount of heat input energy required per hour as a function of the generator's output.

Incremental Cost Curve

By multiplying the input-output curve by the cost of the fuel in \$/MBTU, one obtains the cost curve for the unit in \$/hr. By taking the derivative of the cost curve, one obtains the incremental cost curve, which indicates the marginal cost of the unit: the cost of producing one more MW of power at that unit.

Generator/Load Cost Models

The Generator/Load Cost Models option from the Case Information menu allows you to choose to view detailed information on generator cost curves or load benefit curves. This cost information can be very important when solving an [Optimal Power Flow](#) or [Security Constrained Optimal Power Flow](#) solution.

The Generator/Load Cost Models option has four submenu options to choose from:

[Generator Cubic Cost Models](#)

[Generator Piecewise Linear Cost Models](#)

[All Generator Cost Models](#)

[All Load Benefit Models](#)

Generator Cost Models Display

The Generator Cost Models Display presents detailed cost information for each generator in the case, regardless of what type of cost curve has been entered for the generator. The Generator Cost Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cost Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear and cubic cost model display, select **Network > Generators > Cost Curves All** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can be assigned to an area different than the area to which its terminal bus is assigned.

ID

Single character ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

Min MW

Minimum MW output of the generator.

Max MW

Maximum MW output of the generator.

Cost Model

The type of model this generator is currently using. Can be Cubic, Piecewise Linear or None.

IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

These fields will be disabled unless the Cost Model type is set to Cubic.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Variable O&M

Operations and Maintenance costs for the generator.

This field will be disabled unless the Cost Model type is set to Cubic.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Fixed Cost

The fixed operating cost of the generator.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.

A requirement of the piecewise linear cost curve is that it must be convex, meaning the next MWh Price must be higher than the previous MWh Price.

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

Generator Cubic Cost Display

The Generator Cubic Cost Display presents detailed cost information for each generator in the case set to use a cubic cost model. The Generator Cubic Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cubic Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator cubic cost display, select **Network > Generators > Cost Curves Cubic** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is [different](#) than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Variable O&M

Operations and Maintenance costs for the generator.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Cost \$/Hr

Operating cost for the generator in \$/hr.

IC

Incremental cost to produce an additional MWh. This can be expressed as $dC(P_{gi})/dP_{gi}$, where C denotes the generator's cost of operation in \$/hr and P_{gi} expresses the current MW output of the unit. In a lossless system, the incremental dispatch is equal to the generator's lambda value.

LossSens

Area loss sensitivity - This field is **only calculated when the generator's area is on economic dispatch control**. This field specifies the incremental change in area losses if this generator were to produce one more MW, **with the excess generation absorbed by the system slack**. This may be expressed as $\partial P_{loss}/\partial P_{gi}$. The loss sensitivity is used in calculating the generator's lambda value for the economic dispatch activity.

Generator MW Marg. Cost

The marginal cost of the generator supplying an additional MW of power to the system.

Generator Piecewise Linear Cost Display

The Generator Piecewise Linear Cost Display presents detailed cost information for each generator in the case set to use a piecewise linear cost model. The Generator Piecewise Linear Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Piecewise Linear Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear cost display, select **Network > Generators > Cost Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the generator is attached.

Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is different than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the generator. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power output of the generator.

Min MW

Minimum MW output of the generator.

Max MW

Maximum MW output of the generator.

Fuel Cost

The fuel cost of the type of fuel for the generator.

Fuel Type

An informational field that can be set to the type of fuel the generator uses.

Unit Type

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

Fixed Cost

The fixed operating cost of the generator.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.

Load Display

The Load Display presents data describing each load in the case. The Load Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated loads. The toolbar also affords the opportunity to insert new loads into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the load's terminal bus. You can also sort the load information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To show the load display, select **Network > Loads** in the [Model Explorer](#).

The load display contains the following fields by default:

Number, Name

Number and name of bus to which the load is attached.

ID

Two-character ID used to distinguish multiple loads at the same bus; '1' by default.

Status

Either Closed if the load is connect to its bus, or Open if it is not. You can click on this field to toggle its value. If the load is open, the entire load record is dimmed.

MW, Mvar, MVA

Total real, reactive, and complex power for the load. Loads may be voltage dependent. The total load is the sum of the constant power, constant current, and constant impedance components. See [Load Information](#) and [Load Modeling](#) for more information.

S MW, S MVAR, I MW, I MVR, Z MW, Z MVR

These six fields describe the composition of the load at the bus assuming 1 pu bus voltage. The SMW and SMVAR fields indicate the constant power portion of the load, the component that does not vary with bus voltage magnitude. The IMW and IMVR fields express the constant current part of the load, which varies in proportion to the bus voltage magnitude. Finally, ZMW and ZMVR indicate the constant impedance portion of the load, which varies with the square of the voltage. The sum of the SMW, IMW, and ZMW fields yields the base MW load at the bus (assuming 1 pu voltage), and the sum of the SMVR, IMVR, and ZMVR fields provides the base MVR load at the bus (assuming 1 pu voltage). Please see [Load Modeling](#) for more details on how bus load is modeled.

Load Benefit Models Display

The Load Benefit Models Display presents detailed cost information for each load in the case set to use a piecewise linear benefit model. The Load Benefit Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated loads. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the load's terminal bus. You can also sort the load records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Load Benefit Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new load into the case or to delete an existing load.

To show the generator piecewise linear cost display, select **Network > Loads > Benefit Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

Number, Name

Number and name of bus to which the load is attached.

Area Name of Gen

Name of the area to which the load belongs. The load can belong to an area which is different than the area of which its terminal bus is a member.

ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

Status

The Open / Closed status of the load. This is a toggleable field.

AGC

Designates whether the generator's real power output is governed by automatic generation control, since effectively dispatching a load can be viewed as dispatching negative generation. If the AGC field is set to *Yes*, the load is on automatic generation control (AGC). When a load is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

Gen MW

Current real power demand of the load.

Min MW

Minimum MW demand of the load.

Max MW

Maximum MW demand of the load.

Fixed Benefit

The fixed benefit of the load.

Benefit Model

The type of model this load is currently using. Can be either Piecewise Linear or None.

MWh Break x, MWh Price x

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW demand value of the load. The MWh Price value is the corresponding marginal benefit of extracting an additional MW of load at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal benefit of the load from that break point location to the maximum demand of the load.

A requirement of the piecewise linear benefit curve is that it must be concave, meaning the next MWh Price must be lower than the previous MWh Price. In other words, as more load is supplied, the less the benefit it is providing. This is how a load can be dispatched along with generation, according to marginal costs (and marginal benefits.)

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

Line and Transformer Display

The Line/Transformer Display presents data describing each transmission line and transformer in the case. The Line/Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated branches. The local menu also affords the opportunity to insert new lines or transformers into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for each branch's terminal buses. You can also sort the line and transformer information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line/Transformer Display according to the constraints of the [area/zone/owner filters](#).

To show the line/transformer display, select **Network > Branches Input** or **Network > Branches State** from the [Model Explorer](#).

The contents of the display depends upon the application's operating mode.

The line/transformer display shows the following fields by default:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *from bus*.

To Bus Number and Name

To Bus number and name. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *to bus*.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Status

The service status of the branch. This field is toggleable.

Branch Device Type

Identifier for the type of branch. Options are: Line, Series Cap, Breaker, Load Break Disconnect, Disconnect, ZBR, Fuse, and Ground Disconnect. The type of branch can be changed through this field except that a line cannot be changed from or to a Transformer. To make that change, use the [Branch Options dialog](#) in Edit mode. The type of branch becomes very important when using [Integrated Topology Processing](#). Breakers and Load Break Disconnects are treated as special switching devices which can be controlled automatically through special actions, such as "Open with Breakers" and "Close with Breakers".

Xfrmr

Yes or no field signifying if the branch is a transformer or transmission line. This field cannot be changed.

MW From, Mvar From, MVA From (State display default)

Real, reactive, and complex power flowing into the line at the *from bus*.

In versions of Simulator prior to version 14, these values were always calculated based on the present voltage at a bus and other branch parameters each time they were displayed. This means that a user could change the bus voltage, or impedance, and see a corresponding change in the line flow values. These values are no longer calculated each time that they are displayed. They are updated following a power flow solution, loading in a case, following an update to the system due to OPF control changes, and after creating an equivalent. The exception to this is that when loading in a PowerWorld Binary (*.pwb) file and the flows are stored in the file, the stored flows will be retained instead of updating them based on the present system state. The MW and Mvar values can be set to user-specified values using an auxiliary file or pasting from Excel. This will allow the contouring of flows based on some external source.

Four additional fields have been added that will update each time they are displayed. This means that they will be updated following a voltage change for a bus or other relevant parameter (i.e. impedance) change for a branch. These fields take on the functionality of the flow fields prior to version 14 and are calculated each time that they are displayed. These fields are MW From (Calculated), MW To (Calculated), Mvar From (Calculated), and Mvar To (Calculated). These fields cannot be changed by the user.

Lim MVA, % of MVA Limit (Max) (State display default)

The current MVA limit of the branch, and the amount of the actual flow as a percentage of the MVA limit.

MW Loss, Mvar Loss (State display default)

Real and reactive power losses on the transmission line or transformer. Since reactive power losses include the effect of line charging, the reactive power losses may be negative.

R, X, B (Input display default)

Indicates the branch's resistance, reactance, line charging susceptance in per unit on the system base.

Lim A MVA, Lim B MVA, Lim C MVA (Input display default)

Identifies the first three limit settings. Five additional limits can be set, for which the columns can be added using the Display/Column options from the local popup menu. All limits are expressed in MVA.

See [Transformer Display](#) for viewing transformer specific fields.

Merge Line Terminals

A transmission line can be removed from a load flow case without losing the electrical connectivity of the line's two terminal buses by merging the line terminals into a single bus. The command to merge line terminals can be performed in two ways, and can only be invoked when Simulator is in Edit mode.

First, a [MergeLineTerminals](#) script command can be invoked in Script mode or from an auxiliary file. The only parameter of the script command is a filter parameter, which must be populated with either the name of an [advanced filter](#) (with the name in quotation marks) or with the text SELECTED (no quotation marks.) If an advanced filter is given, Simulator will find all transmission lines that meet the advanced filter definition, and will individually merge the line terminals of each line one at a time. Simulator will retain the From bus information as the new merged bus information for each line, with one exception. If a line was removed by merging its terminals that was the first or last segment of a [multi-section line](#), the multi-section line terminal bus information will always be retained as the new merged bus.

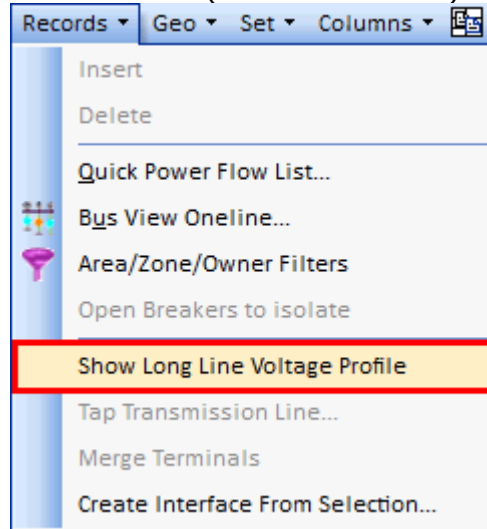
The second way a line or lines can be chosen to have their line terminal buses merged is to select a set of lines in the branches table of the [model explorer](#), and to then right-click in the table and bring up the popup menu. The option to Merge Terminals is located under the Branch Records submenu.

Long Line Voltage Profile

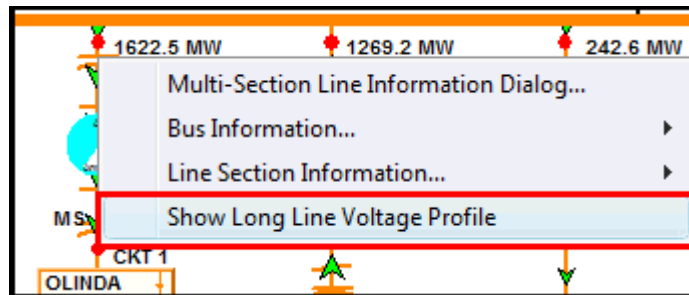
Normally when using a power flow solution tool, you only look at the voltages at the terminal buses of transmission lines. However, for transmission lines which have large shunt charging susceptance (B) terms, the voltages at intermediate points of the transmission line can become much higher (or even lower) than the terminal bus values. To see these voltages directly, Simulator provides a tool for showing the Long Line Voltage Profile of a transmission line (or multi-section line).

You may show the voltage profile along the length of a transmission line in two ways. First on any oneline diagram (including the Bus View), you may right-click on a transmission line and choose **Show Long Line Voltage Profile**.

Second on the Records menu of any case information display showing transmission branches, there will be an option to **Show Long Line Voltage Profile**. If you select multiple transmission lines, then the dialog will show the voltage profile on each transmission line.

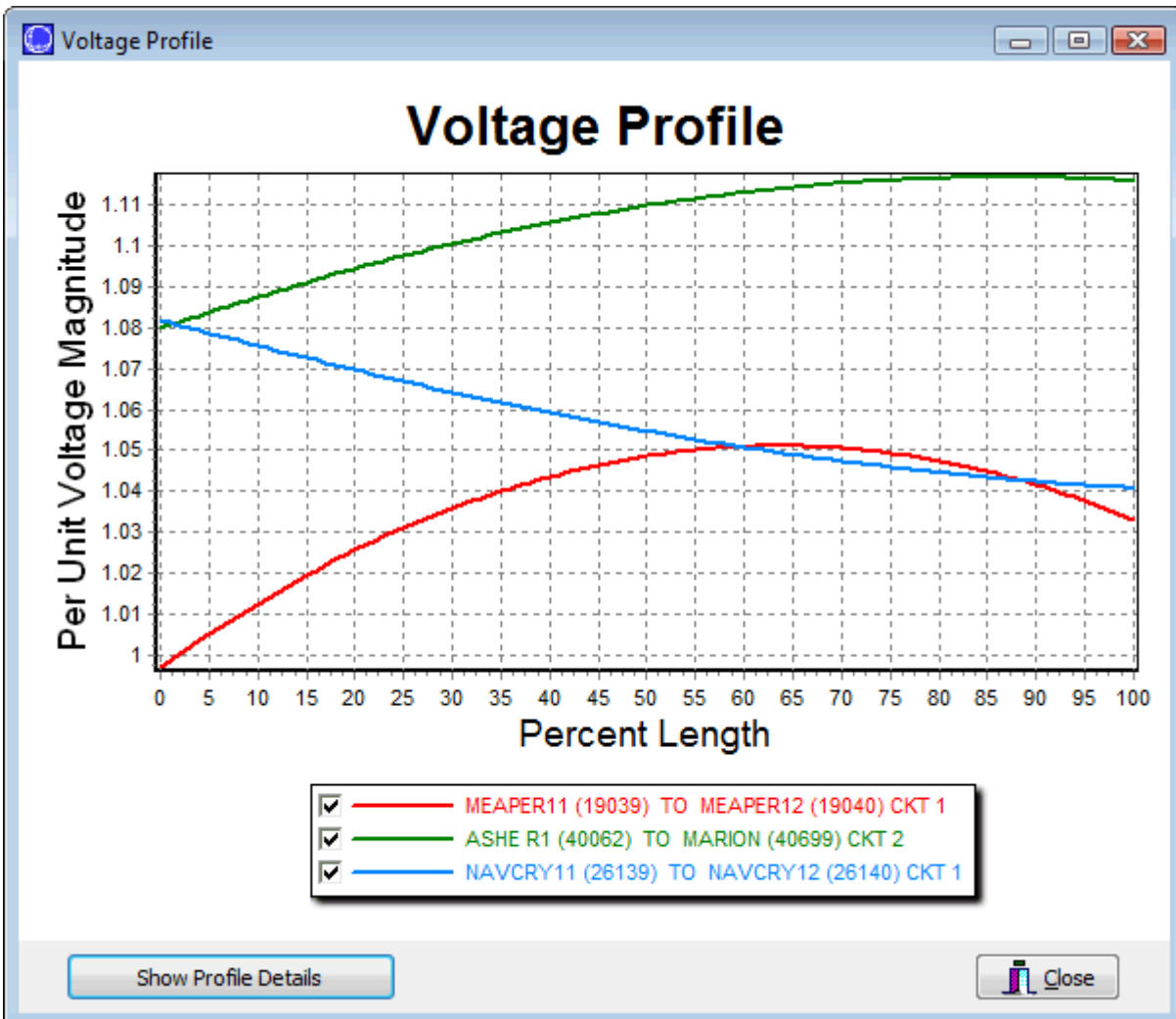


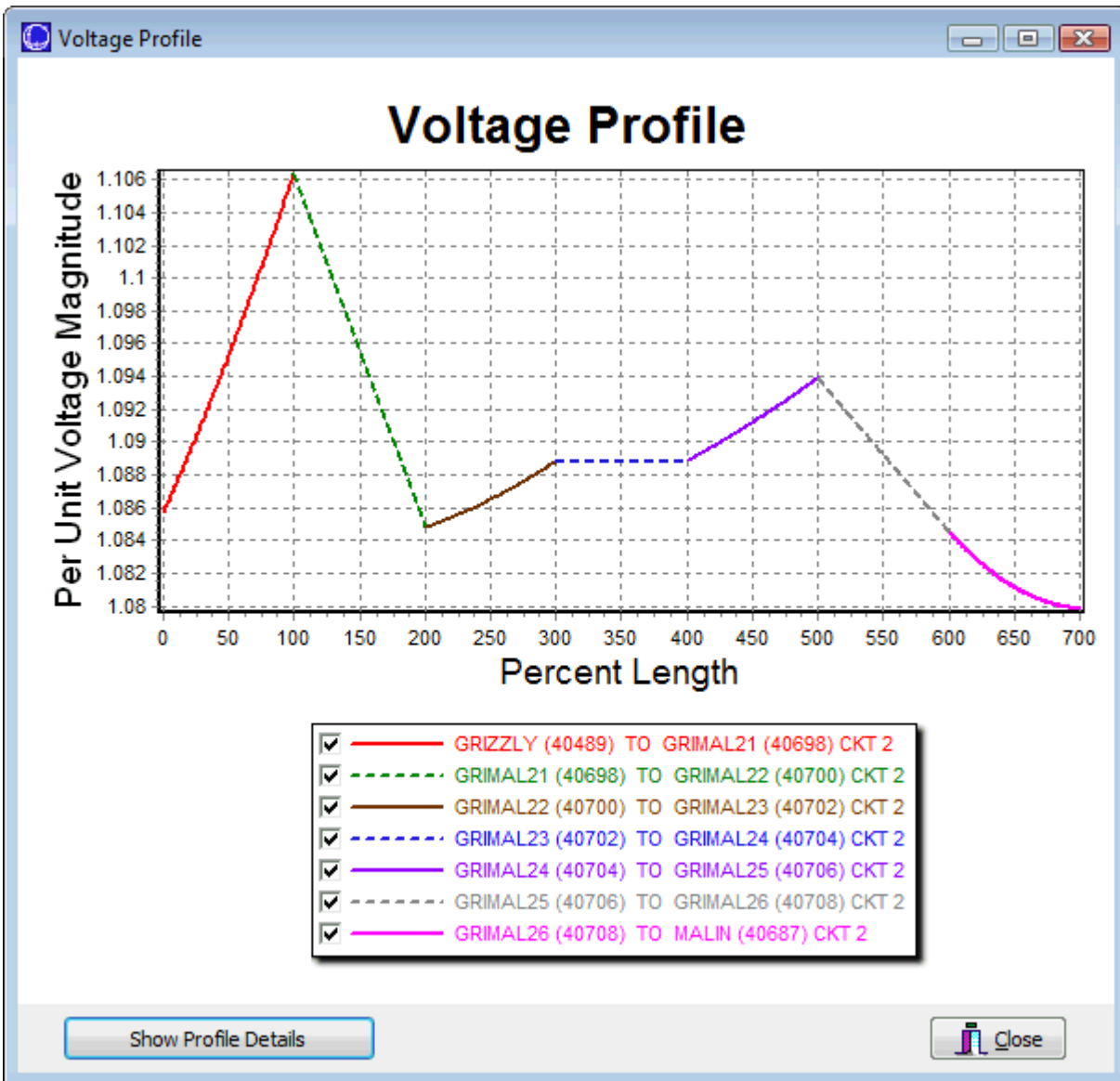
After you have chosen to show the voltage profile, a dialog will appear showing a graph of the voltage profile for each line selected. There will be a legend with check boxes that allow you to hide or show particular curves. The horizontal axis will represent the percent length of the transmission line. The vertical axis will represent the per unit voltage at each particular point along the transmission line. A per unit voltage is calculated at 101 points along the transmission line starting at 0%, 1%, ... up to 100% of the length of the transmission line. The voltage shown at 0% represents the voltage at the FROM bus of the transmission line. The voltage shown at 100% represents the voltage at the TO bus of the transmission line. This is shown in the figure below.



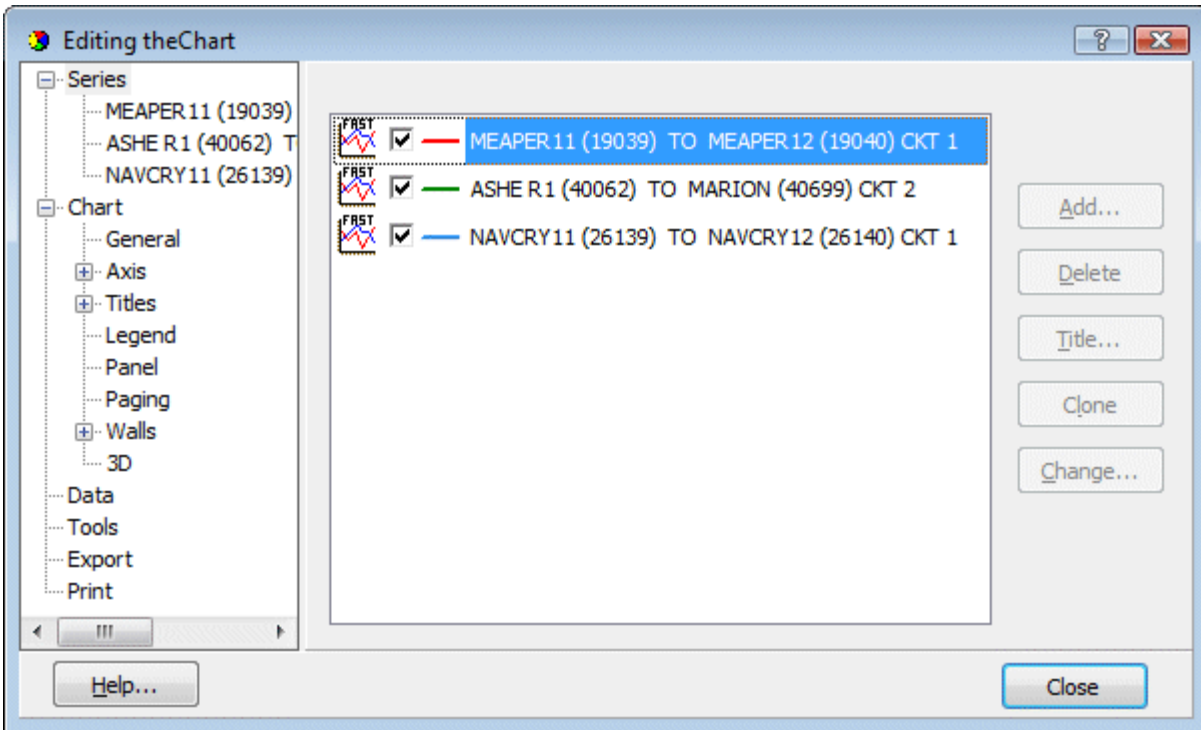
When showing show the voltage profile across a multi-section line, each line section is represented separately on the plot and is shown in one of the "100 ranges" on the x-axis. For instance the third line section will appear between

200% and 300%. Also series caps and transformers are shown as dotted lines on these plots. This is shown in the second figure below.

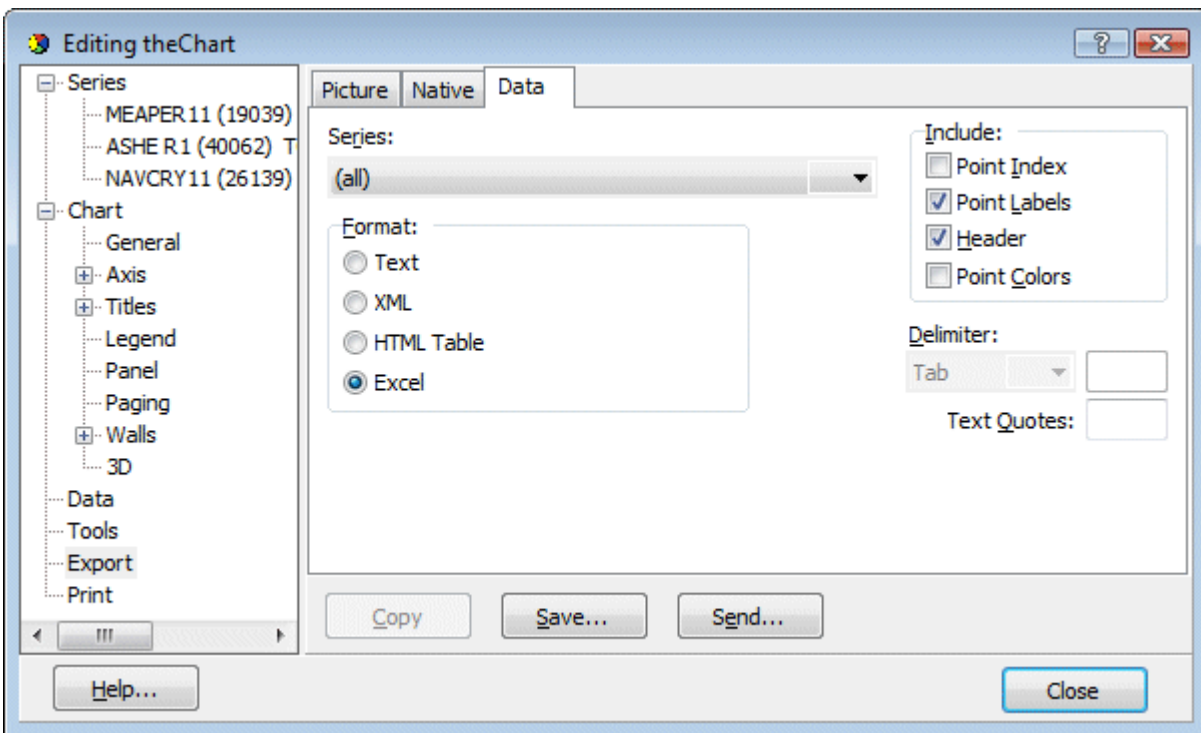




To get exact details of what the voltage profile can be shown by clicking on the **Show Profile Details** button. This will display a dialog as shown next.



Of particular interest on the dialog that appears is the ability to export this data to Excel or text. This can be done by going to Export and then choosing the Data tab as shown next. Clicking on the **Save** button will then save the data to a CSV file.



You may also play with this dialog to see all the possibilities for customizing your plots as well as printing them.

Multi-Section Lines Display

To show this display select **Aggregations > Multi-Section Lines** in the [Model Explorer](#).

The Multi-Section Line Display lists the multi-section lines that exist in the case. The Multi-Section Line Display is used to functionally group a number of transmission lines together. They are usually used to model very long transmission lines that require the use of multiple individual transmission line information to be modeled accurately. Simulator then treats each multi-section line as a single device with regard to line status. That is, changing the status of one line in the record changes the status of the other lines in the record as well.

Each multi-section line record consists of the "from" bus, one or more "dummy" buses and the "to" bus. Each dummy bus must have only two lines connected to it, each of which are members of the multi-section line record. See [Multi-Section Line Information](#) for details.

The Multi-Section Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its multi-section line information. The local menu also affords the opportunity to insert new multi-section lines into the model or to delete existing ones when the application operates in the Edit Mode. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the line's terminal buses. You can also sort the multi-section line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Multi-Section Line Display according to the constraints of the [area/zone/owner filters](#).

While in Edit mode, the local menu allows three options that are specific to multi-section lines. The options are found under the **Multi-Section Line records** menu item.

Show Long Line Voltage Profile

For more information on this see the [Long Line Voltage Profile](#) topic.

Merge Sections

Select this option to eliminate the selected multi-section line record and designate the section lines as single line records.

Renumber Dummy Buses

This option allows renumbering/renaming the dummy buses that are part of a multi-section line by loading a file that has the new designations. The file containing the renumbering should be a text file in the following format:

```
frombus, tobus, circuit // this identifies the multi-section line
// frombus and tobus can either be bus numbers or name_nomkV combinations
dummybusnumber1, dummybusname1
dummybusnumber2, dummybusname2
frombus, tobus, circuit // this identifies another multi-section line
dummybusnumber1, dummybusname1
dummybusnumber2, dummybusname2
dummybusnumber3, dummybusname3
```

The file can either be space or comma delimited. A double slash can be used for a comment. A line with three entries is assumed to be a multi-section line and any two entry lines following that are assumed to be the dummy bus identifiers belonging to that multi-section line. Dummy buses are read for a multi-section line until another three entry line is found, meaning that a new multi-section line has been specified. Dummy buses are identified starting at the from bus of the multi-

section line going to the to bus and that is the way that they should be entered in the file. If a multi-section line matching the given identifiers cannot be found, all dummy buses after that multi-section line record are ignored until a new multi-section line specification is found. If an incorrect number of dummy buses is entered for a multi-section line, none of the dummy buses will be updated. If the a dummy bus number is specified that matches an existing bus that is not another dummy bus, no changes will be made for that dummy bus. If a dummy bus number is specified that matches an existing bus that is another dummy bus, the other dummy bus will be assigned to a new bus number and the current dummy bus will be assigned to the number specified in the file.

The display contains the following fields by default:

NumberFrom, NameFrom

Number and name of the multi-section lines *from bus*. Right-clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *from bus*.

NumberTo, NameTo

Number and name of the multi-section lines *to bus*. Right-clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *to bus*.

Circuit

Two-character circuit identifier for the multi-section line.

Sections

Number of individual lines within the multi-section line record.

Allow Mixed Status

This value defaults to NO and means that all branches in the multi-section line must have the same status. Setting this value to YES essentially means that the multi-section line is disabled so that opening/closing a branch in the multi-section line will no longer automatically open or close the other lines in the record.

Status

Current status of the record. Note, the status is *Closed* only if **all** the lines in the record are closed, and is *Open* only if **all** the lines in the record are open. Otherwise the status is *Mixed*.

Transformer Display

The Transformer Display identifies all transformers in the case. The data presented in the Transformer Display supplements the data presented in the [Line/Transformer Display](#) by presenting transformer-specific information. Consult the Line/Transformer Display for the transformer flows.

The Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated transformer. You can find a specific transformer using the names or numbers of its terminal buses, and you can learn more about a particular transformer's terminal buses by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can delete an existing transformer from the case. You can also sort the transformer information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Transformers Controls** in the [Model Explorer](#).

The Transformer Display contains the following fields by default:

From Bus Number and Name

From Bus number and name. The *From Bus* is the tapped side. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *From Bus* from the local menu.

To Bus Number and Name

To Bus number and name. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *To Bus* from the local menu.

Circuit

Two-character identifier used to distinguish between multiple transformers joining the same two buses.

Status

The service status of the transformer. This field is toggleable.

Type

Type of transformer. Possible values include

Fixed The tap positions are fixed

LTC The tap ratio changes to regulate bus voltage

Mvar The tap ratio changes to regulate reactive power flow

Phase The phase angle changes to regulate real power flow

Tap/Phase

Indicates the tap ratio for LTC and fixed transformers and the phase shift angle in degrees for phase-shifting transformers.

XF Auto

If the value of this field is *Yes*, the transformer will automatically change its tap or phase angle to keep the regulated value within the specified regulation range, provided that the *Auto XF* field of its associated area is set to *Yes* and transformer tap/phase control has not been disabled for the entire case. The *Auto XF* field of individual areas is set from the [Area Display](#), and case-wide transformer control can be set from the Power Flow Solution tab of the [Simulator Options](#) Dialog. Click on this field to toggle its values.

Reg Bus

For an LTC transformer, this is the number of the bus whose voltage is controlled by the transformer. For phase shifting transformers, the real power is always controlled at the tapped bus.

Reg Value

For an LTC transformer, this is the present per unit voltage at the regulation bus. For a phase shifting transformer, this is the present real power flow through the transformer measured on the "from" (tapped) side.

Reg Error

The error is the difference between the regulated value and the respective limit of the regulation range specified by Reg Min and Reg Max. If the regulated value is within the regulation range, then the error is zero. The error is negative if the regulated value falls below the regulation range, and it is positive if the regulated value exceeds the regulation range.

Reg Min, Reg Max

Minimum and maximum values for the regulation range. For LTC transformers, these fields represent per unit voltage at the regulated bus. For phase shifting transformers, these fields represent actual MW flow through the transformer measured on the "from" (tapped) side. Because transformers use discrete control, the maximum regulation value must be somewhat greater than the minimum value.

Tap Min, Tap Max

For LTC transformers, these fields specify the minimum and maximum tap ranges for the transformer. For phase shifting transformers, these fields specify the minimum and maximum phase shift angle in degrees.

Step Size

The per unit step size for tap changing transformers. This step size is usually determined by dividing the total range of transformer operation by the number of tap positions available.

Three Winding Transformer Display

The Three Winding Transformer Display shows information about all the three winding transformer devices in the case. Three winding transformers are modeled in Simulator as a grouping of two winding transformers, connected at a common midpoint or "Star" bus. The Three Winding Transformer Display is a way to view all the terminal connection points for the three winding transformers and the power delivered at each of the terminals.

Depending on how the three winding transformer was loaded into Simulator, the star bus may be dynamically created for the three winding transformer, with Simulator choosing the star bus number and name. If the star bus numbers and names of three winding transformers need to be changed, an option exists to load the new star bus numbers and names from a file. This option can be selected from the Three-Winding Transformer display in the [Model Explorer](#) by right-clicking on the table and choosing **3W Transformer Records > Renummer Star Buses** while in [edit mode](#). The format for the text file is one record per row, with six fields for each record. The first four fields are primary bus number, secondary bus number, tertiary bus number, and circuit ID. These fields identify for which three winding transformer the star bus is being changed for that record. The last two fields for each record will be the new star bus number and new star bus name. If the new bus number specified is already in use in the case, the record will be ignored and the star bus will not be renumbered for that three winding transformer record. Note that the first three fields can be specified as the "name_kv" of each of the three winding transformer terminal buses instead of the bus numbers.

The Three Winding Transformer Display is a class of [Case Information Display](#), and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated three winding transformers. When in Edit Mode, you can define new three winding transformers using the **Insert** option, or delete existing three winding transformers using **Delete**. You can also sort the three winding transformer information by clicking on the heading of the field by which you want to sort.

To show this display, open the [model explorer](#) and click on **Network > Three Winding Transformers**.

This display contains the following fields by default:

Pri Bus Num, Sec Bus Num, Ter Bus Num

These are the primary winding, secondary winding and tertiary winding terminal bus numbers for the three winding transformer connections.

Circuit

The circuit identifier for the three winding transformer.

Pri, Sec and Ter MW and MVar

These fields display the real and reactive power delivered at each of the three winding transformer terminals.

Star Bus Number and Name

These two fields provide the number and name of the internal node star bus used in the three winding transformer equivalent model.

While in Edit mode, the local menu allows an option that is specific to three-winding transformers. The option is found on the **3W Transformers records >** menu item:

Renumber Star Buses...

This option allows renumbering the star buses of three-winding transformers to user-specified values. This is done via a text file that is loaded with the new designations. The format of the text file is the following:

Primary bus, Secondary bus, Tertiary bus, circuit, new star bus number, new star bus name

Either the bus number or name_nominal kV (secondary key field) combination may be used to identify the buses. Each bus may be identified using either method even for the same three-winding transformer. Use double slashes (//) to indicate a comment in the file. Any text past the double slashes will be ignored. The file may be either space or comma delimited. If the tertiary bus does not exist, this entry should be either a 0 or empty string enclosed in double quotes. If the new star bus number already exists and is the star bus of another three-winding transformer, the star bus number of the other three-winding transformer will be reassigned and the current transformer will take the specified number. If the specified new star bus number already exists and is not taken by another star bus, the three-winding transformer will not be updated with the specified new star bus number.

Line Shunts Display

The Line Shunts Display identifies all line shunt devices in the case. Line shunts are similar to [switched shunts](#) in that they are used in power systems either to inject additional MVR into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). However instead of being connected to a bus directly, they are instead shunt connected to one of the terminals of a transmission line. As a result they can only be in-service if the actual transmission line is in-service. Also, Simulator also does not allow the automatic regulation of a bus voltage using Line Shunts as is possible using a switched shunt. Simulator does however allow line shunts to be individually taken in and out of service. This must be done manually however using the status field of the line shunt.

The Line Shunts Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated line shunts. You can find a specific line shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can insert a new line shunt into the case or delete an existing shunt. You can also sort the line shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line Shunts Display according to the constraints of the [area/zone/owner filters](#).

To call up the Line Shunts Display, click **Network > Line Shunts** in the [Model Explorer](#).

The Line Shunts Display contains the following fields by default:

From Number and To Number

Numbers of from and to bus of the transmission branch to which the line shunt is attached. Use the right menu options to inspect either the [Quick Power Flow List](#) or the [Bus View Display](#) for the terminal buses.

Circuit

Circuit identifier for the transmission branch to which the line shunt is attached.

Bus Number Located At

Either the From or To Bus number. Specified the end of the transmission branch to which the line shunt is attached.

ID

A circuit identifier for the actual line shunt. This is needed because there can be multiple line shunts at the same end of the same transmission line.

G, B

Indicates the line shunts nominal conductance and susceptance in per unit on the system base.

Status

Line shunts can be individually taken in and out of service. Do this by changing the status field to Open or Closed.

DC Lines Display

The DC Line Display presents data describing each dc line in the case. The DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated dc lines. You can also sort the dc line records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the DC Line Records Display according to the constraints of the [area/zone/owner filters](#).

To show the dc line records display, select **Network > DC Transmission Lines** in the [Model Explorer](#), and click on the DC Lines tab.

By default, the dc line records display contains the following fields:

Number

Each dc line must be assigned a unique number, typically between 1 and 40.

Rect AC Number, Rect AC Name, Rect MW, Rect Mvar

Number and name of the rectifier bus, and the real and reactive power flow from the rectifier into the dc line. You may right-click on any of the rectifier-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the rectifier bus.

Inv AC Number, Inv AC Name, Inv MW, Inv Mvar

Number and name of the inverter bus, and the real and reactive power flow from the inverter into the dc line. You may right-click on any of the inverter-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the inverter bus.

Control Mode

Specifies how flow on the dc line is controlled. If this field is set to *Power*, then the line's MW flow is the control parameter. If the control mode is defined as *Current*, then the line's current is the control parameter.

Setpoint Magnitude, Setpoint Location

Specifies the initial value of the control parameter, which is expressed either in MW or in amps depending on the dc line's control mode. The Setpoint Location indicates if the setpoint should be set at the Inverter or Rectifier end of the DC line.

Set kV

Specifies the voltage of the dc line in kV.

Multi-Terminal DC Record Display

The Multi-terminal DC Line Display presents data describing each multi-terminal DC line in the case. The Multi-Terminal DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated records. The local menu also affords the opportunity to insert multi-terminal DC line records into the model or to delete existing ones.

To show the Multi-Terminal DC Line display, select **Network > DC Lines** in the [Model Explorer](#), and click on the Multi-terminal DC Lines tab of the transformer case information display.

The Multi-terminal DC Line display shows the following fields by default:

Number

The record number of each multi-terminal DC record.

Num Conv

The total number of converters in each multi-terminal DC network.

Num Buses

The total number of DC buses in each multi-terminal DC network.

Num Lines

The total number of DC lines in each multi-terminal DC network.

Mode

The control mode of the multi-terminal DC network: 0 is Blocked control, 1 is Power control, and 2 is Current control.

V. Cont. Bus

The number of the AC converter bus at which the DC voltage is controlled.

Switched Shunt Display

The Switched Shunt Display identifies all [switched shunt](#) devices in the case. Switched shunts are used in power systems either to inject additional Mvar into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). They may also be used to regulate bus voltage within some specified range.

The Switched Shunt Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated switched shunts. You can find a specific switched shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by looking at the [Bus View](#). When in Edit Mode, you can insert a new shunt into the case or delete an existing shunt. You can also sort the switched shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Switched Shunt Display, select **Network > Switched Shunts** in the [Model Explorer](#).

The Switched Shunt Display contains the following fields by default:

Number, Name

Number and name of terminal bus to which the switched shunt is attached. Note that multiple switched shunts are allowed at a bus, however only one switched shunt can have a Control Mode which is not fixed. Use the right menu options to inspect either the [Quick Power Flow List](#) or the [Bus View Display](#) for the terminal bus.

ID

Circuit identifier for the switched shunt device.

Reg Bus Num.

Number of the bus whose voltage is regulated by the switched shunt. If the regulation bus is the same as the terminal bus, this field is blank.

Status

The service status of the switched shunt. This field is toggleable.

Control Mode

Control Mode for the switched shunt. A switched shunt may operate either as *Fixed* if its reactive value is to be held constant, as *Discrete* if its reactive output is controlled automatically in discrete steps to regulate its terminal voltage, or as *Continuous* if its reactive injection is allowed to vary over a continuous spectrum of values to regulate its terminal voltage. A shunt will be switched either discretely or continuously to regulate its terminal bus' voltage between its high and low voltage limits provided the *Auto Shunt* field of its corresponding area is set to *Yes* and switched shunt control has not been disabled for the entire case. You may set the value of the *Auto Shunt* field from the [Area Display](#), and you can control case-wide enforcement of shunt switching from the Power Flow Solution tab of the [Simulator Options](#) Dialog. This Control Mode field is toggleable.

See [Switched Shunt Control](#) for more information on how switched shunt output is automatically adjusted during power flow solutions.

Regulates

Can be set to regulate bus voltage, generator Mvar, or wind generator Mvar output.

Actual Mvar

The reactive power currently supplied by the switched shunt.

Volt High

Per unit high-voltage limit for the regulation range. It is important for discrete shunt control that Volt High exceed Volt Low by a nontrivial amount; otherwise, the output of the shunt may oscillate during the Power Flow Solution.

Volt Low

Per unit low-voltage limit for the regulation range.

Reg Volt

Actual per unit voltage at the regulated bus. When Control Mode is either *Discrete* or *Continuous*, this voltage should be between Volt Low and Volt High.

Deviation

Deviation of the regulated bus' actual per unit voltage from the desired regulation voltage. If the actual voltage is within the regulation range, this field is zero. If the voltage is greater than Volt High, the deviation is positive. It is negative if the actual voltage is less than Volt Low.

Nominal Mvar

The Nominal Mvar field gives the initial amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

Max Mvar, Min Mvar

The maximum and minimum Mvar range for the switched shunt.

Interface Display

The Interface Display is used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines (AC and DC) and/or transformers, 2) total tie-lines between two adjacent areas, 3) total tie-lines between two adjacent zones, 4) Generators and Loads, 5) Injection Groups, 6) multi-section lines, 7) other interfaces and 8) Contingency Actions . Interface information is useful because secure power system operation often requires that the flow on such groups be less than some limit value. For example, interface information could be used as "proxies" for other types of security constraints, such as voltage or transient stability limitations. Another major potential use for interfaces is in the [Optimal Power Flow](#) solution to monitor flows on groups of devices, such as a set of transmission lines, generation leaving a plant, load demanded in a certain region, etc. Interface information can also be extremely useful for summarizing the flows occurring on a large network. Interface flows can be monitored using the [Limit Monitoring Settings Display](#).

The Interface Display presents more detailed information for each interface that has been defined for the case. The Interface Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated interfaces. The local menu also affords the opportunity to insert new interface definitions either singly (**Insert**) or as a group ([Automatic Insertion](#)) into the model or to delete existing ones. In addition, PowerWorld has also added options for reading NERC flowgate files ([Load NERC Flowgates](#)) and writing NERC flowgate files ([Save NERC Flowgates](#)) to the local menu.

You can sort the interface information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Interfaces** in the [Model Explorer](#).

Interface information can be saved in the "*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The interface Display is divided into two tabs: **Interface** and **Interface Elements**.

Interface Elements Tab

The **Interfaces Elements tab** has a list of tabs which lists each types of element that can belong to an interface. There is a check-box below in the middle of the display which allows you to **Only show elements that belong to at least one interface**. As you select an element in the top half of the display, the bottom half of the display will list each interface which contains the selected element. Again, these provide a nice way to look through the elements that make up each interface.

Interface Tab

The **Interfaces tab** has a list of interfaces at the top of the display. When on the Interface tab, in the bottom half of the tab is a set of subtabs which display all the elements of the presently selected interface from the top half of the display. This set of sub tabs starts with the **Elements** tab which lists all the elements of the interface regardless of the type of element. The remainder of the tabs each lists the interface elements of a specific type of interface element such as Branches, Branch Open/Close, DC Lines, Generators, Loads, Injection Groups, Multi-Section Lines and other interfaces. These subtabs provide a nice way to look through the elements that make up each interface. The top half of the Interface display contains the following fields:

Number

Numeric identifier for the interface.

Name

Alphanumeric identifier for the interface (24 characters maximum).

Interface MW

Current MW flow on the interface. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

MW Limit

Current rating for the interface in MW.

Percent

The actual MW flow on the interface as a percentage of the MW limit.

Monitor Direction

The current direction in which the MW flow is being monitored. The possibilities are From - To or To - From.

Monitor Both

Toggle this field to Yes in order to monitor the flow on the interface in both directions, instead of a single direction as designated by the Monitor Direction field.

Lim MW A, Lim MW B, Lim MW C

These three fields display the values of the three possible limits for the interface.

Has Contingency

This is a Yes or No field that will be set to Yes if any of the elements making up part of the interface are at a device limit.

Contingent MW Flow

If the interface defined contains a contingency action, the Contingent MW Flow is the approximated flow amount that would be added to the interface should the contingency occur.

Base MW Flow

The base MW flow is the flow on the interface prior to any considered contingency elements.

Nomogram Display

The Nomogram Display is used to modify and create Nomogram definitions. Nomograms are combinations of two [interfaces](#) for monitoring combined flows on the interfaces concurrently. These Nomogram interfaces will have a limit definition that defines a region of allowed flow on the interfaces, and can be monitored in many tools in Simulator as potential constraints, such as the contingency analysis reporting of interface violations. Another major potential use for nomograms is in the [Optimal Power Flow](#) solution to monitor the flow on a pair of interfaces. Nomogram information can also be extremely useful for summarizing the flows on a pair of interfaces whose operation and allowed flow are closely tied together. Nomogram flows can be monitored using the [Limit Monitoring Settings Display](#).

The Nomogram Display presents more detailed information for each nomogram that has been defined for the case. The Nomogram Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated nomograms.

You can sort the nomogram information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Nomograms** in the Model Explorer.

Nomogram information can be saved in the "*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The Nomogram display contains the following fields:

Name

Alphanumeric identifier for the nomogram.

Int A Flow

Current MW flow on the first interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

Int B Flow

Current MW flow on the second interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

% Limit, Max Nomo-interface

The actual MW flow on the nomogram as a percentage of the combined nomogram limit.

Monitor

Indicates whether or not the nomogram should be monitored either for violations or as a constraint in the [Optimal Power Flow](#).

Limit Group

The name of the limit group the nomogram is a member of for [limit monitoring](#).

Injection Group Display

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

Name

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

Number of Gens

Identifies the number of generators contained in the injection group.

% MW Gen ParFac

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

Number of Loads

Identifies the number of loads contained in the injection group.

% MW Load ParFac

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

% Mvar Load ParFac

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Number of Shunts

Identifies the number of switched shunts contained in the injection group.

% Mvar Shunt ParFac

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Total MW Injection

Indicates the current total MW injection of all elements in the injection group.

Total Mvar Injection

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

Participation Points Tab

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

Injection Group Dialog

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

Gens

Displays the number of generators contained in the injection group.

Loads

Displays the number of loads contained in the injection group.

Shunts

Displays the number of switched shunts contained in the injection group.

% MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points can be changed by

entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

Participation Point Records Display

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

Point Type

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

Number

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

Name

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

ID

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

AutoCalc?

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

Initial Value

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

SPECIFIED The participation factor was specified as a constant.

MAX GEN INC The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

MAX GEN DEC The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

MAX GEN MW The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

SPECIFIED The participation factor was specified as a constant.

LOAD MW The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

SPECIFIED The participation factor was specified as a constant.

MAX SHUNT INC The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

MAX SHUNT DEC The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

MAX SHUNT MVAR The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

SPECIFIED The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

Injection Group 1 Injection Group 2

Gen 1, ParFac = 50 Gen 5, ParFac = 40

Gen 2, ParFac = 30 Load 1, ParFac = 20

InjectionGroup 2, ParFac = 20 Load 2, ParFac = 20

Total = 100 Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:

Gen 1, Normalized ParFac = $50/100 = 0.5$

Gen 2, Normalized ParFac = $30/100 = 0.3$

Gen 5, Normalized ParFac = $20/100 * 40/80 = 0.1$

Load 1, Normalized ParFac = $20/100 * 20/80 = 0.05$

Load 2, Normalized ParFac = $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

Island Display

The Island Display presents information on the system's islands. An island consists of a group of buses that are electrically connected via ac transmission lines and transformers and thus operate in synchronism with one another. Multiple islands can be connected together via dc transmission lines. Each island requires a slack bus. All systems have at least one island, which may encompass the entire system.

The Island Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information. You can also sort the islands information by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregations > Islands** in the [Model Explorer](#). The display is only available in run mode.

You cannot modify any fields on this display.

The display has the following fields for each island:

Slack Bus Number, Slack Bus Name, Slack Bus Area

Number, name and area of the slack bus for the island. Each island requires at least one slack bus.

Total Buses

Total number of buses in the island.

Energized

Indicates whether the island is connected to a source of power.

Gen MW, Gen Mvar

Total real and reactive generation for the island.

Load MW, Load Mvar

Total real and reactive load for the island.

Scheduled Exports

The power scheduled to be provided by the island to other regions of the system. Because the island is isolated from the rest of the system, this export requirement is currently not met.

DC MW Exports

The power provided by the island to other regions of the system across a DC line.

MW Transactions Display

The MW Transactions Display is a quick way to view the defined base transactions between areas within the load flow case.

The **Matrix of Transactions** display is set up as a matrix of transactions, with the areas listed as both the column and row identifiers. It is important to note that the direction of the transaction is such that the area represented by the **row** is assumed the exporting area, and the area represented by the **column** is assumed the importing area. For example, if there is a 50 MW transaction from area one to area two, you will see a +50 in the matrix in row 1, column 2. However, if you look at row 2, column 1, you will see a – 50. This is because the grid is displaying the EXPORT from area two to area one, but since area two is importing, not exporting, the value is represented as negative.

Each row and column position, with the exception of the diagonal positions (it does not make sense for an area to export to itself), can be directly modified by the user in this information display. As you type a value in one of the matrix positions, Simulator automatically fills the symmetric matrix position with the negative of the value you enter. This makes for a quick and easy location for adding and removing transactions from the case.

Note that you cannot modify **Unspecified** transactions in the MW Transactions display. This makes sense, as the unspecified transactions have only one associated area. To modify unspecified transaction amounts, you must open the [Area Information Dialog](#) for the area you wish to modify, and change the Unspecified transaction amount.

The **List of Transactions** is just another way of showing the transactions between areas. The List of Transactions Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

Note that a special feature is available in the right-click local menu called **Clear Transactions and auto-insert tieline transactions**. By choosing this option all MW transactions in the case will be deleted, and all Unspecified MW transactions for each area will be set to zero. Then new MW transactions will be created between each pair of areas that area directly connect to one another. The amount of the new MW transactions will be set equal to the actual sum of the flow on the tielines between the connected areas.

By default, the List of Transactions display contains the following fields:

Export Area Number, Export Area Name

Exporting area number and alphanumeric identifier.

Other Area Number, Other Area Name

Importing area number and alphanumeric identifier.

MW Transfer

Value of the transfer in MW.

Transaction Dialog

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

Information

Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

Transaction To Area

This is the "to" area for the transaction.

Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

Determine Price in OPF

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

Piecewise Linear Transaction Cost Curve

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

Custom

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.

Owner Data Information Display

The Owner Data display will display the owners sorted by owner number, and show the total number of devices, as well as a breakdown of the number of specific devices. The owner display will also summarize the total device output or demand for the owned devices, such as generator output and load demand.

You can right click on the owner grid to open the local menu, from which you can perform various actions and view an informational regarding the owner selected in the grid. Once an [owner dialog](#) is displayed for a particular owner, you can browse through information on all the owners.

Owner Dialog

The Owner Dialog displays summary information about the devices designated as owned by the selected owner.

The dialog for an individual owner displays each of the elements for that owner, along with the percentage of ownership, in the device pages labeled [Buses](#), [Loads](#), [Generators](#), and [Lines](#).

The dialog also contains a [memo](#) page for making comments or notes about the selected owner.

General Info

Load and Generation

The information in the Load and Generation section provides a summary of the total injections of the owned devices. This includes a total of all the load, generation, and shunt injections owned.

Summary of Owner Objects

This information section simply lists a total number of all owned devices. Note that buses and loads do not currently have fractional ownership. However, generators and transmission lines do have the capability to be partially owned by more than one owner. Therefore it is possible to see fractional ownership amounts for the number of generators or lines.

Generator Costs and OPF Results

This information provides a summary of LMP and cost information determined by running an [Optimal Power Flow](#) solution. The information provided is determined only for devices owned by the selected owner.

Owned Bus Records Display

The Owned Bus Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Bus Display](#). In addition to the information available for viewing from the [Bus Display](#), the Owned Bus Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per bus, so by default the ownership percent is always 100%.

Owned Load Records Display

The Owned Load Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Load Display](#). In addition to the information available for viewing from the [Load Display](#), the Owned Load Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per load, so by default the ownership percentage is always 100%.

Owned Generator Records Display

The Owned Generator Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Generator Display](#). In addition to the information available for viewing from the [Generator Display](#), the Owned Generator Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a generator, right-click on the grid and choose Show Dialog and select the [Owners, Area, Zone tab](#).

Owned Line Records Display

The Owned Line Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Line and Transformer Display](#). In addition to the information available for viewing from the [Line and Transformer Display](#), the Owned Line Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a transmission line, right-click on the grid and choose Show Dialog and select the [Owners tab](#).

Jacobian Display

The Jacobian display is a matrix showing the system Jacobian matrix for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. Be aware that a Jacobian matrix from a large will often exceed the size limitations of an Excel spreadsheet.

Ybus Display

The Ybus display (bus admittance matrix) is a matrix showing the system Ybus for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. However, Excel does have limitations on the number of rows and columns that could quickly be exceeded with a Ybus from a large case.

Bus Information (Edit Mode)

This dialog is used to view/modify information about each bus in the system during Edit Mode. It is very similar in content to its [Run Mode](#) counterpart.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus. You can use the spin button immediately to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

If adding a new bus to the one-line display, this dialog will open after selecting the location of the bus display object. This dialog can be used to add a bus to the one-line for buses that already exist in the power flow case or new buses that should be created. If entering the bus number of a bus that does not already exist in the case, an informational message will appear above the Bus Number field indicating that using this bus number will insert a new bus into the power system data model and not just to the one-line display.

Find By Number

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

Bus Name

Unique alphabetic identifier for the bus.

Find By Name

To find a bus by its name, enter the bus name into the Bus Name field (case insensitive) and then click this button.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Nominal Voltage

Nominal voltage of the bus in kV.

Labels

Clicking on this button will open the [Label Manager Dialog](#) listing all of the labels assigned for the selected bus.

Area Number

Number of the bus' area. Each bus must be associated with an area record. If a bus is specified as belonging to an area that does not already exist in the model, the new area is created.

Area Name

Alphabetic identifier for the bus' area. If the area already exists, you do not need to enter this value.

Zone Number

Number of the bus' zone, between 1 and 2,147,483,647 (equals 2^{31} minus 1). Each bus must be associated with a zone record. If a bus is specified as belonging to a zone that does not already exist in the model, the new zone is created. Zones provide a useful mechanism for breaking up a large system. Buses can be assigned to zones independent of their area assignments. Thus, a single area could contain multiple zones, or a single zone could span multiple areas. You can use the [Zone](#)

[Dialog](#) to list the buses in a particular zone and easily move a group of buses from one zone to another.

Zone Name

Alphabetic identifier for the bus' zone. If the zone already exists, you do not need to enter this value.

Owner Number

The number of the bus' owner.

Owner Name

The name of the bus' owner.

Substation Number

The number of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

Substation Name

The name of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

OK, Save, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the **Find By Number** command to edit additional buses. **Cancel** closes the dialog without saving any changes.

Bus Information

Voltage (pu), Angle (degrees)

Current per-unit voltage magnitude and angle for the bus. If you are inserting a new bus into an existing system, **you should not change the initial per unit voltage values**. Rather, when you first switch to Run Mode, Simulator will estimate the voltage magnitude and angle at the new buses in such a way as to reduce the initial mismatches. **This automatic estimation is only available if you have not modified the voltage in any way.** Select **Bus Voltage Regulator Devices** to view the individual devices regulating the voltage for the bus. Selecting this button displays the [Bus Voltage Regulating Devices Dialog](#).

System Slack Bus

Check only if the bus should be modeled as a system slack bus. Each case requires at least one slack bus. Simulator can also dynamically determine slack buses as described in the [Advanced Power Flow Solution Options](#).

Display

Orientation

Set the orientation of the bus. The current choices are right, left, up or down. For the current bus object shapes of Rectangle and Ellipse, right and left are analogous to horizontal, up and down are analogous to vertical. Additional shapes for buses may be available in the future for which right, left, up and down may have more specific impact on the appearance of the bus object.

Shape

Sets the shape of the bus object.

Width

Specifies the horizontal axis of an elliptical bus or horizontal side length of a rectangular bus.

Size

Specifies the vertical axis of an elliptical bus or the vertical side length of a rectangular bus.

Scale Width with Size

When this option is checked, changing the size will cause the width to automatically adjust to keep the same ratio of width to size. To adjust the width independent of the size, uncheck this option or adjust the width separately after adjusting the size.

Link to New Bus

If you have right clicked on a bus and opened the bus information dialog, you could change the bus number in the **Bus Number** field and press this button to force the bus object on the diagram to link to the new bus number and information in the load flow data.

Attached Devices

Load Summary Information

Displays the net MW and Mvar load at the bus. You cannot change either of these fields from this display. Select the Add or Edit Bus Load Records to view the individual load records for the bus via the [Load Dialog](#).

Shunt Admittance

The real (G) and reactive (B) components of shunt compensation at the bus, expressed in MW and MVR, respectively.

Geography

The Geography page of the dialog provides information on the geographic location of the bus on the currently active online diagram. Also it provides a conversion tool between [Lat/Lon and UTM Coordinates](#).

Fault Parameters

This tab is only visible when viewing the bus information for a bus with attached load. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#).

Custom

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Bus Field Information

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to change bus device properties.

The Bus Fields Information Dialog can be used to modify the properties of individual bus fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

This option is currently not used for bus fields.

Maintain Constant Load Power Factor

This option is only used for certain fields, such as load fields and generator fields. It is not used currently for any bus fields.

Include Suffix

If this checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Anchored

If this checkbox is checked, the bus field is [anchored](#) to its associated bus, which means that it will move with the bus.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Type of Field

Used to determine the type of bus field to show. The following choices are available:

Bus Name Name of the bus

Bus Number Bus number (from 1 to 2,147,483,647 (equals 2^{31} minus 1))

Bus Voltage Bus voltage magnitude

Bus Angle Bus voltage angle in degrees

MW Marginal Cost Bus MW marginal cost in \$/MW•hr; available only with OPF

Mvar Marginal Cost Bus Mvar marginal cost in \$/Mvar•hr; available only with OPF

MW Loss Sensitivity Increase in losses for 1 MW of generation over the nominal

Select a Field Choose from any defined Bus fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Shortest Path Between Elements

To access the Shortest Path Between Elements go to the [Tools](#) ribbon tab, and choose **Connections > Determine Shortest Path Between...** from the [Other Tools](#) ribbon group. The Determine Shortest Path Between dialog provides a way to find the shortest electrical pathway between two parts of the power system.

After clicking the **Calculate** button, the shortest path of AC branches will be determined with the first bus being inside the Start Element and the last bus being inside the End Element. Options described below allow the user to choose the Start Element, End Element, Distance Measure for each branch, and branches that are allowed to be traversed. The results will be displayed in a case information display at the bottom of the dialog showing the sequence of buses that are traversed between the start and end elements.

The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

End Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the ending element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch.

Lines to Process

Regardless of the Distance Measure above, you can choose which branches are allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed.

Bus Voltage Regulating Devices Dialog

The Bus Voltage Regulating Devices Dialog displays summary information about the devices regulating by the selected bus. The regulating devices can be Generators, Switched Shunts and Transformers.

Also the Bus Voltage Regulating Devices Dialog displays a [Case Information Display](#) for the regulating devices. In addition to the information available about the [Generators](#), [Switched Shunts](#) and [Transformers](#) Display, the Bus Voltage Regulating Devices Dialogs Display also shows and allows to modify the Regulating Max and Min per unit voltage, the Device Voltage Target, the Transformer (if any) Regulation Target Type and Device Voltage Target High.

Substation Information (Edit Mode)

The Substation Dialog is used in the [Edit Mode](#) to view information about a substation and to move one or more buses from one substation to another. (See [Substation Information \(Run Mode\)](#) for help on the corresponding [Run Mode](#) version.) To view this dialog, first select **Aggregation > Substations** from the [Model Explorer](#) to view the [Substations Records](#) Display. Then, right-click on the desired substation record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

Substation Number

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

Substation Name and ID

Two alphanumeric identifiers for the substation. You can use these fields to change the substation's name or ID, provided you click either Save or OK.

Find By Number

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

Find By Name

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

Find By Sub ID

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

Find...

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

Labels

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

Buses

This table lists all of the buses in the substation. Number, name, voltage, area number and name, and zone number and name are shown for each bus. This table can be used to move buses to a different substation. Select the bus or buses you would like to move with the mouse. Then, enter the Destination Substation Number, which is the substation to which you want to move the selected buses. You may enter a substation number that does not already exist, too, so that the buses will be moved to a brand new substation. In this case, be sure to provide the new substation a name and ID, as well. Finally, click the Move Selected Bus(s) to Destination Substation button to implement the move.

Display Options

The Display Options tab of the Substation Information dialog is only visible if an open diagram contains an object representing the substation. This tab allows you to choose the general appearance

of the substation object. Use the Width and Height fields to set the **width** and **height** of the substation object. The **Shape** field allows you to choose what shape the substation object will take.

By clicking on **Link to New Object**, the Choose Object selector will appear, allowing the user to select another substation in the load flow data to be linked to this particular graphical object.

The **Substation Layout Online, URL or Command** field allows you to specify the same of the Simulator online diagram (pwd) file that should be automatically opened if you click on the substation object on the diagram in run mode. If no online is specified, Simulator will search for a default name of the format AreaName_SubstationName. If a diagram cannot be found, then Simulator will not attempt to open any online diagrams when the substation is clicked. Optionally, a HTTP URL can be entered in the form <http://example.com>. Also, a line command can be entered, such as *Excel.exe c:\file.xls*.

Substation Generators, Loads, and Switched Shunts Tables

These tables list all of the generators, loads, and switched shunts in the substation. Number, name, ID, status, and additional fields for each type of device are shown.

Custom

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom fields can also be entered for storage with the selected substation.

Geography

Displays geographic information about the location of the substation, in Latitude and Longitude. When showing the dialog by right-clicking on a [substation online display object](#) from a online diagram, then a button will also appear that states **Copy Longitude/Latitude from Online Location**. This button will then determine the longitude/latitude of the center of the substation object on the online diagram and copy these values into the Substation Data Record. Also it provides a conversion tool between [Lat/Lon and UTM Coordinates](#).

OK, Save, Cancel

OK saves any changes to the substation name or ID, and closes the dialog.

Save saves any changes to the substation name or ID, but does not close the dialog.

Cancel closes the dialog ignoring any changes.

Substation Field Options

Substation field objects are used to show different values associated with substations. This dialog is used to view and modify the parameters associated with these fields.

Substation Number

Select the number of the substation for which you are inserting or viewing information of a substation field.

Find...

If you do not know the exact substation you are looking for, you can click this button to open the [advanced search engine](#).

Substation Name

The name of the currently selected substation.

Substation ID

The substation ID number.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Anchored

Toggle the anchoring of the field to the substation object of which it is associated.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of super area field to show. The following choices are available:

Substation Name, Substation Number

Name or number of the selected substation.

Substation ID

ID string for the selected substation.

Max Nominal Voltage

Displays the nominal voltage of the highest nominal voltage bus in the substation.

Substation Load MW, Substation Load Mvar

Total Load MW or MVAR in the substation.

Substation Gen MW, Substation Gen MVAR

Total Generator MW or MVAR in the substation.

Select a Field

Select this option and click **Find Field...** to display the value of any other substation specific field.

Generator Information (Edit Mode)

This dialog is used to view and modify the parameters associated with each generator in the system. It can also be used to insert new generators and sometimes to delete existing generators.

The Edit Mode version of the Generator Information Dialog is almost identical to the [Run Mode](#) version.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [area/zone/owner filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with valid [area/zone/owner filters](#).

ID

Two-character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click this button.

Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click this button.

Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

Area Name

Alphabetic identifier for the terminal bus' area.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There are six additional areas of information on this dialog for specific aspects of generation:

[Display Information](#)

[Power and Voltage Control](#)

[Costs](#)

[Fault Parameters](#)

[Owner, Area, Zone, Sub](#)

[Custom](#)

[Stability](#)

Generator Field Information

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

The Generator Fields Information Dialog can be used to modify the properties of individual generator fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

ID

ID of the generator associated with the field. Generator ID are two-character alphanumeric fields.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Generator fields can be used not only to show various fields associated with generation devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the generator field. When the up spin button is clicked, the generator field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the generator field value is decreased by the same amount.

This field is only used for fields of the following types: Setpoint Gen MW, Setpoint Gen Mvar, and Gen Setpoint Voltage. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the generator field on the oneline.

Rotation Angle in Degrees

The rotation angle at which the text field should be displayed.

Anchored

If the *Anchored* checkbox is checked, the generator field is [anchored](#) to its associated generator, which means that it will move with the generator.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of generator field to show. The following choices are available:

Gen MW Output MW generation

Gen Mvar Output Mvar generation

Gen AGC Status AGC status of generator; status can be toggled in Simulator

Gen AVR Status AVR status of generator; status can be toggled in Simulator

Gen Setpoint Voltage Voltage of generator on bus

Select a Field Choose from any of the different generator fields

Select **OK** to save changes and to close the dialog, or click Cancel to close the dialog without saving your changes.

Generator Options: Display

This information is located on the Display Information tab of the [Generator Information Dialog](#).

Display Size

The size of the generator.

Scale Width with Size

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

Display Width

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Generator

Links the object to a different generator in the data.

Rotor Shape

Several shapes are available for the rotor symbol. The default is the "dog bone" shape, but options are also available for symbols that represent the fuel type of the unit. There is no link between the Fuel Type field that can be specified with a generator and the rotor symbol. The rotor shape must be specified independently of the Fuel Type field.

Fill Rotor Symbol with Color 2, Fill with Color 2

Check the **Fill Rotor Symbol with Color 2** box to use the color specified in the **Fill with Color 2** field to fill the rotor symbol with a different color than the generator background color.

Generator Options: Power and Voltage Control

This information is located on the [Generator Information Dialog](#).

Power Control

The Power Control grouping fields are used to show/change the values associated with the real power output of the generator.

MW Output

Current real power output of the generator.

Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control anytime you manually change its output. You could then place the generator back on AGC control if you wish it to participate in an area generation dispatch.

Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

Voltage Control

The Voltage Control grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

Available for AVR

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to

maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

Use Capability Curve

If checked, then the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

Regulated Bus Number

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** in the [Model Explorer](#) to view the [Remotely Regulated Buses](#) display, which identifies all buses that are being remotely regulated.

SetPoint Voltage

Specifies the desired per unit setpoint voltage value the generator is to regulate at the regulated bus. The regulated bus need not be the generator terminal bus.

Remote Reg %

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100. If the total value is different from 100%, then all the regulation factors are normalized to obtain a percentage of regulation.

Wind Control Mode and Power Factor

The Wind Control Mode effects how generator Mvar limits are treated and can be set to either *None*, *Boundary Power Factor* or *Constant Power Factor*.

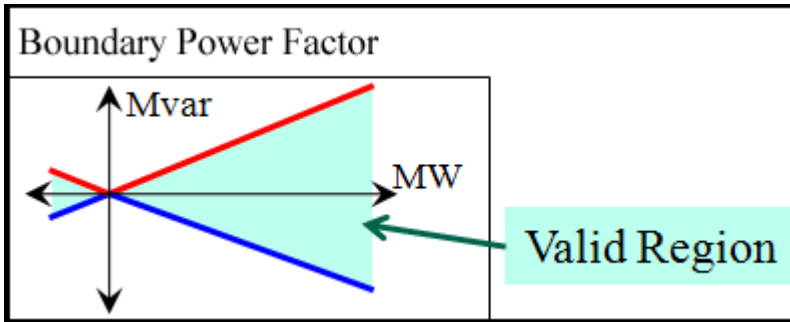
When the mode is *None*, then the generator behaves using the standard settings of the **Min and Max Mvar Output** and **Capability Curve** described above.

For *Follow Min Mvar Capability Curve* mode, the **Mvar output** of the generator is determined by a lookup from the capability curve's Min Mvar value. Essentially the **Max Mvar Output** and the **Min Mvar Output** are then made equal to this value. This provides the ability to make a generator's **Mvar output** any piece-wise linear function of the **MW output**.

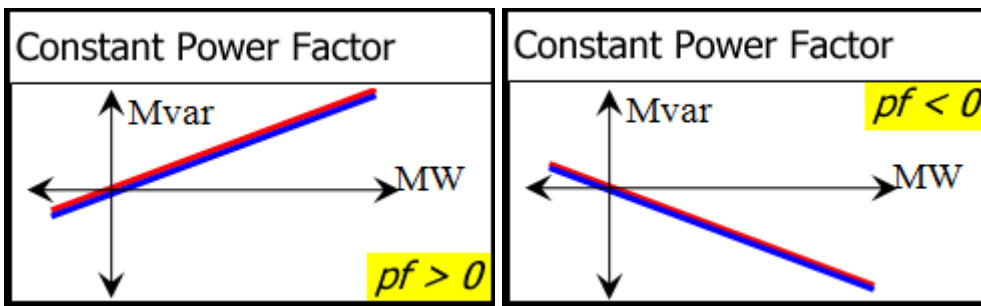
For both the *Boundary* and *Constant* modes, the Mvar limit magnitudes are determined from the actual **MW Output** and the **Wind Control Mode Power Factor** value.

$$\text{Magnitude} = \text{MWOutput} * \tan(\arccos(\text{Power Factor}))$$

For *Boundary* mode, the **Max Mvar Output** is positive and the **Min Mvar Output** is negative. This provides a boundary under which the Mvar must operate.



For *Constant* mode, **Max Mvar Output** and the **Min Mvar Output** are made the same with a positive **Wind Control Mode Power Factor** meaning the limits have the same sign as the actual **MW Output**, and a negative **Wind Control Mode Power Factor** meaning the limits are the opposite sign as the actual **MW Output**, thus the **Mvar output** is a function of **MW Output** and **Power Factor**.



Generator Options: Costs

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

Output Cost Model

Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form $C(P_{gi}) = (d \cdot P_{gi}^3 + c \cdot P_{gi}^2 + b \cdot P_{gi}) \cdot (\text{fuel cost}) + \text{fixed costs}$ (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

Bid Scale / Shift

Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Marginal Cost (run mode only)

Shows the marginal cost of producing real power at the generator at its current output level, $dC_i(P_{gi})/dP_{gi}$.

ED/OPF Cost (run mode only)

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

Unscaled Cost (run mode only)

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Generator Options: Fault Parameters

The parameters on this tab are used when running a [fault analysis](#) study.

Generator MVA Base

The assumed MVA base for the generator. This value is used when calculating fault analysis values for the internal generator parameters.

Neutral Grounded

Check this check-box if the generator has the neutral grounded.

Generator Step Transformer

The resistance, reactance and tap setting for the generator step-up transformer, if one is being modeled internally with the generator. By default, no internal transformer model is assumed.

Internal Impedance

These fields represent the internal impedance of the generator for all three sequences. By default, all three values are initially the same as the load flow internal impedance of the generator. All three sets of values can be modified, either manually or by loading values from an external file using the [Fault Analysis Dialog](#).

Neutral-to-Ground Impedance

Neutral-to-ground impedance for the generator. These values get implemented with the zero sequence admittance matrix. Note that the neutral-to-ground impedance will not be used, even if specified, if the original model for the generator implicitly models the generator step-up transformer. This is because the implicitly modeled transformer is assumed to have a delta winding on the generator side of the transformer, which isolates the generator from the rest of the zero sequence network.

Generator Options: Owners, Area, Zone, Sub

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

Owners

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

Area Number, Area Name

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

Generator Cost Description

The cost associated with operating a generator varies according to the output of the generator, with the general rule that getting more power out of a generator costs more. In Simulator, there are two options for modeling generator cost. The first employs the common cubic relationship

$$C_i(P_{gi}) = \text{IndFixedCost} + (a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3) * \text{fuel cost } \$/\text{Hour}$$

where P_{gi} is the output of the generator at bus i in MW. IndFixedCost is the fuel cost independent fixed cost, and therefore is not multiplied by the fuel cost. The value a_i is the fuel cost dependent fixed cost, and is therefore multiplied by the fuel cost. The values b_i , c_i , and d_i are used to model the generator's input-output (I/O) curve. The I/O curve specifies the relationship between how much heat must be input to the generator (expressed in MBtu per hour) and its resulting MW output. Normally, the cubic coefficients remain constant for a generator. The last term in the equation is the fuel cost, expressed in $\$/\text{MBtu}$. This value varies depending on the fuel used in a generator. Typical values would be $\$ 1.25/\text{MBtu}$ for coal and $\$ 2/\text{Mbtu}$ for natural gas. The resultant equation is known as the fuel-cost curve. The value of the fixed costs, b_i , c_i , d_i , and fuel cost can be viewed and modified using the [Generator Information Dialog](#).

Simulator can also model generator costs using a piecewise linear model consisting of pairs of MW output and incremental cost ($\$/\text{MWhr}$) of generation, along with a fixed cost. These piecewise linear curves must be convex curves, meaning the marginal cost of the current MW break point must be higher than the previous MW break point.

Such curves can be defined using the [Generator Information Dialog](#) or by loading data from [generator cost data](#) files.

Set Generator Participation Factors

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its area's load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

Max MW Rating of Generator The participation factor for each generator is set to the generator's maximum MW capability.

Difference Between Max and Current Output The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

Constant Value of The participation factor for each generator is set to the same hard-coded value.

File The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check

the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

Generator Reactive Power Capability Curve

The reactive power output of most generators depends on the real power output of the generator. This dependence is expressed using a reactive capability curve. Simulator models the reactive capability curve using a piecewise linear approximation. The reactive capability curve is modified on the [generator dialog](#) and can be saved/loaded using the [Generator Reactive Capability Curve Auxiliary files](#).

Modeling a Reactive Power Capability Curve From the Generator Dialog

- Make sure the **Use Capability Curve** checkbox is checked.
- In the table at the bottom of the dialog, prescribe the reactive capability curve using up to 50 points. The points should be ordered by MW, in numerically increasing order. At each point specify the MW value, the minimum reactive power value in Mvar, and the maximum reactive power value in Mvar. The first point should correspond to the minimum MW output of the generator while the last point should correspond to the maximum MW output although there is not a strict requirement that the generator's minimum and maximum output be specified in the curve. If the MW output of the generator is less than the MW value of the first point specified for the curve, the first curve point will be used to determine the minimum and maximum Mvar. If the MW output of the generator is greater than the MW value of the last point specified for the curve, the last curve point will be used to determine the minimum and maximum Mvar.
- To insert a new point, click on the desired column, and then right-click to display the table's local menu. Select Insert Point.
- To remove a point, click on the desired column, and then right-click to display the table's local menu. Select Delete Point.
- When finished be sure to select **Save** to save your modifications.

In the Run Mode, you can view the reactive power capability curve graphically by right-clicking on the generator to display its submenu and then selecting **Reactive Capability Curve**.

Load Information (Edit Mode)

This dialog is used to view and modify the parameters associated with each load in the system. It can also be used to insert new loads and sometimes to delete existing loads. It is nearly identical in structure to its [Run Mode](#) counterpart.

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with valid [area/zone/owner filters](#). You can use the spin button to cycle through the list of load buses.

When you insert objects graphically, the Bus Number and Bus Name fields are usually set automatically to the bus upon which you placed the object.

Bus Name

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with valid [area/zone/owner filters](#).

ID

Two-character ID used to distinguish multiple loads at a bus. By default, the load ID is equal to "1 ." An identifier of "99" is used to indicate an equivalent load.

Find By Number

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

Find By Name

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

Find...

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

Labels

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

Area Number, Area Name

Number and name of the area the load is a member of.

Zone Number, Zone Name

Number and name of the zone the load is a member of.

Substation Number, Substation Name

Number and name of the substation the load and the terminal bus are members of.

Owner Number, Owner Name

Number and name of the owner the load is a member of. If the load owner is the same as the terminal bus owner, the **Same Owner as Terminal Bus** box will be checked. Loads DO NOT have to be owned by the same owner as the terminal bus.

OK, Save, Delete, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional loads.

Delete deletes the current load. **Cancel** closes the dialog without saving your changes.

Specific load information can be found on the following pages of the Load Options dialog:

[Load Information](#)

[OPF Load Dispatch](#)

[Custom](#)

[Stability](#)

Load Options: Load Information

This page of the Load Options dialog contains information on the load magnitude and display settings.

MW and Mvar Value Fields

The MW and Mvar Value fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields can be modified by the user.

Display Size

Size of the load.

Scale Width with Size

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

Display Width

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Load

Links the object to a different load record in the data.

Load Options: OPF Load Dispatch

This tab of the Load Options dialog contains settings for allowing the load to be included as an OPF Control. The load(s) can then be dispatched in the OPF algorithm according to the assigned costs.

Benefit Model

Benefit Model

If this field is set to none, the load will not be dispatchable in the OPF solution. If the option is set to Piecewise Linear, the load is dispatchable during the OPF, according to the following fields.

Min. and Max. MW Output

Minimum and maximum load MW demand for OPF dispatch.

Available for AGC

If checked, the load will be available for redispatch during the OPF routine.

Fixed Benefit

Value of the load benefit at minimum demand.

Piece-wise Linear Benefit Curve

This table allows you to specify pairs of MW demand levels and corresponding load benefit values, which in turn define the starting points and slopes of the piece-wise linear benefit curve segments. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Load Field Information

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

The Load Fields Information Dialog can be used to modify the properties of individual load fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

ID

ID of the load associated with the field. Load ID fields are two characters in length.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Load fields can be used not only to show various fields associated with load devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the load field. When the up spin button is clicked, the load field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the load field value is decreased by the same amount.

This field is only used for fields of the following types: Load MW and Load Mvar. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the load field on the oneline.

Note that the **Maintain Constant Load Power Factor** option will allow you to specify a Delta per Mouse-click for the MW load, and when the MW value is changed in run mode, the MVAR load will also change in such a way as to keep the power factor of the load constant.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Anchored

If the *Anchored* checkbox is checked, the load field is [anchored](#) to its associated load, which means that it will move with the load.

Rotation Angle in Degrees

The angle at which the text is placed on the diagram, in degrees.

Type of Field

Used to determine the type of load field to show. The following choices are available:

Load MW MW load

Load Mvar Mvar load

Select a Field Choose from any of the different load fields

Select **OK** to save changes and to close the dialog, or click Cancel to close the dialog without saving your changes.

Load Modeling

Each load can be modeled as having voltage variation. The voltage variation is modeled using the Base Load Model fields on the [Load Information](#) dialog. Thus the actual real and reactive value of each load is determined using the following equation:

$$MW = \text{Load MW Multiplier} * (SMW + IMW * V + ZMW * V * V)$$

$$Mvar = \text{Load Mvar Multiplier} * (SMvar + IMvar * V + ZMvar * V * V)$$

where

MW current real power load in MW

Mvar current reactive power load in Mvar

SMW constant power MW value

Smvar constant power Mvar value

IMW constant current MW value (assuming 1.0 per unit voltage)

Imvar constant current Mvar value (assuming 1.0 per unit voltage)

ZMW constant impedance MW value (assuming 1.0 per unit voltage)

Zmvar constant impedance Mvar value (assuming 1.0 per unit voltage)

V per unit bus voltage magnitude

Load MW Multiplier = (Area MW Multiplier) * (Zone MW Multiplier)

Load Mvar Multiplier = (Area Mvar Multiplier) * (Zone Mvar Multiplier)

The Area Multipliers scale all of the loads in an area. The Zone Multipliers scale all of the loads in a zone. These are static values that do not vary over time. If a time varying schedule of loads is required, the [Time Step Simulation](#) tool should be used instead of the static multipliers.

Branch Options (Edit Mode)

The Branch Options dialog is used to view and modify the parameters associated with each transmission line and transformer in the system. You can also insert new transmission lines and delete existing transmission lines from this dialog.

The Edit Mode version of this dialog is very similar in content to its [Run Mode](#) counterpart.

The Branch Options dialog has the following fields:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side.

To Bus Number and Name

To Bus number and name.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Find By Numbers

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

Find By Names

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, then the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

Default Owner (Same as From Bus)

Read-only check-box that indicates whether the line's owner is the same than the from bus' owner.

From and To Bus Area Name

Name of control area in which each terminal bus is located.

From and To Bus Nominal kV

Nominal voltage level of each terminal bus, in kV.

Labels

Clicking on this button will open the [Label Manager Dialog](#) listing all the [labels](#) assigned for the selected branch.

OK, Save, Delete, Cancel, and Help

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the **Find By Number** command to edit additional transmission lines. **Delete** deletes the current branch. The Delete button is not visible if editing a branch that was accessed via the online diagram. **Cancel** closes the dialog but does not save any changes. **Help** opens this help topic.

Display

See [Branch Options: Display](#).

Parameters

See [Branch Options: Parameters](#).

Transformer Control

This tab is only visible for transformer objects. See [Transformer Control](#) for details on modeling either LTC or phase shifting transformers.

Series Capacitor

This tab is only visible for series capacitor objects. See [Series Capacitor](#) for details on modeling series capacitors.

Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes a grounded wye to grounded wye transformer, which has the same model as a transmission line. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

Owner, Area, Zone, Sub

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the branch's owner is the same as the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the branch options dialog.

The area, zone, and substation to which the From and To buses belong, are also shown.

Custom

This page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the branch. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the branch. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Stability

This tab is only visible with the [Transient Stability Add-On](#) tool. Any branch-specific transient stability modeling information is contained on this tab. For more information see the [Transient Stability Data: Object Dialogs](#).

Branch Options: Display

The Display tab of the [Branch Options](#) dialog is available when opening the dialog by right-clicking on a display object.

Pixel Thickness

Thickness of the display object in pixels.

Anchored

If checked, the object is anchored to its terminal bus, which means that it will move when you move the terminal bus. See [Anchored Objects](#) for details.

Link to New Line

Use the **Link to New Line** button to create a new line corresponding to the entries you have made in the dialog. This button performs the same function as pressing **Save**. Note that adding a new line to the case in this way does not add a transmission line to the online display; the new line is present only in the model. You may then add the newly modeled line to the online diagram in the usual way (such as select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

Symbol Segment

Only visible for transformer and series capacitor objects. This field specifies which "segment" of the branch contains the transformer or capacitor symbol. A segment constitutes a section of the line between vertex points of the line object, and are numbered starting at the from bus.

Symbol Size

Only visible for transformer and series capacitor objects. Specifies the size (width) of the transformer or capacitor symbol on the branch.

Symbol Percent Length

The distance or "length" of the symbol on the segment of the branch which contains it.

Show Detailed Line Vertices

When checking this option, a new group of information appears, showing the x,y coordinates of every vertex of the line. Additionally to being able to edit those locations, it is possible to shift or scale all the values with the buttons **Shift All Values** and **Scale All Values**, respectively.

Branch Options: Parameters

The Parameters tab of the [Branch Options](#) dialog displays the common parameters required for defining a transmission branch.

Status

Current status of the device. Either *Open* or *Closed*.

Length

If the length of the line is known, it can be entered here in miles for informational purposes.

Per Unit Impedance Parameters

The series resistance and reactance (R and X), the total charging susceptance (that is, B, not B/2) and the shunt conductance (G) of the device (in per unit). In the case of transformers, the magnetizing conductance and susceptance can also be specified.

Line Shunts

Click this button to view the [Line Shunt](#) Dialog. This dialog is used to only view or modify the values of the line shunts. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

MVA Limits

Ratings for the transmission line or transformer in MVA. Eight different limits are allowed.

Calculate Impedances

This will display a pop-up menu with the following two options:

...From Per Distance Impedances

Clicking this item will open the [Line Per Unit Impedance Calculator dialog](#), which can be used to convert actual impedance and current limits to per unit impedance and MVA limits, and vice versa.

...From Conductor Type and Tower Configuration

Clicking this item will open the [Transmission Line Parameter Calculator](#) dialog, which can be used to compute per unit impedance values, based on a conductor type and a tower configuration.

Convert Line to Transformer

Clicking this button turns the currently selected transmission line into a transformer, making the transformer specific fields available.

Line Shunts Information

The Line Shunts Information dialog is used to modify the parameters of transmission line shunts. The modeling of the line shunts is explained below.

Line Shunts

Line shunts are included in the model as admittance-to-ground values at either the From end or the To end of a transmission line. These values can represent many things, such as shunt-to-ground capacitors, reactors, zigzag (or grounding) transformers, or equivalenced system values. Line shunt values are entered in per unit, with a positive B corresponding to capacitors and a negative B corresponding to reactors. Mathematically, the line shunts are included in the algorithm in the same manner that line charging capacitance is included using the Pi model.

Line shunt values can be modified from the [Branch Options](#) dialog or [Branch Information Dialog](#).

Line Per Unit Impedance Calculator Dialog

The Line Per Unit Impedance Calculator Dialog allows to convert actual impedance and current limit values to per unit impedance and MVA limit values, and vice versa.

The dialog has the following elements:

Actual Impedance and Current Limits

This part of the dialog shows all the impedance related values in Ohms/length-unit as well as the transmission line limits specified in Amps. If any of these values is modified, the corresponding per unit or MVA value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

Line Length

This value indicates the length of the line in miles or kilometers, depending on the units selected in **Length Units**. The option **When changing convert** is used to convert the values when the length value is modified. The option **PU/MVA -->** indicates that the actual impedance and current limits will be converted to per unit impedance and MVA limits when the length value is changes. The option **<-- Electrical** specifies that the per unit impedance and MVA limits will be converted to actual impedance and current limits when the length value is modified.

Length Units

This option indicates the length units. Choices are miles and kilometers. When this parameter is changed, the user will be prompted to confirm to convert the actual impedance and current limits, as well as the line length values, from the old units to the new units. If the answer is positive, then the actual impedance and current limits, and the line length will be the same but they will be expressed in the new units selected. If the answer is negative, the values will not change numerically but they still will be expressed in the new units selected.

System Base Values

The system base values show the power base, the voltage base, and the impedance base. These values can not be modified in this dialog.

Per Unit Impedance and MVA Limits

This part of the dialog shows all the impedance related values in per unit as well as the transmission line limits specified in MVA. If any of these values is modified, the corresponding actual or Amps value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

Conductor Type

Shows the conductor type selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

Tower Configuration

Shows the tower configuration selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

Calculate PU Impedances From Conductor Type and Tower Configuration

Clicking this button will open the [Transmission Line Parameter Calculator](#) form.

Transmission Line Parameter Calculator

The transmission line parameter calculator is a tool designated to compute characteristic line parameters given the type of the conductor and the configuration of a three-phase overhead transmission line.

The parameters computed are the resistance R , reactance X , susceptance B , and conductance G . These values are computed as distributed (per unit of distance), lumped or total (for a specific line distance), and in per-unit.

Calculations

The following controls are part of the calculations section:

Parameters Calculation

This section is to enter the necessary data to compute the characteristic line parameters that are shown in the Results panel.

Input Data

Conductor Type: This is the combo box that lists all the conductor types available in the Conductors table. To add, remove or edit a specific conductor and its characteristics, see Conductor Type section below.

Tower Configuration: This combo box lists all the tower configurations available in the Tower Configurations table. To add, remove or edit a specific tower configuration, please go to the Tower Configuration section below.

Line Length: This is the value of the distance of the transmission line. The units are miles when using English system, or kilometers when using the Metric (SI) system.

Line Length Units: The line length units specify the measurement system to use when entering the line length. The options are English system or Metric (SI) system. The final and intermediate results will also be shown in the system specified here.

Power Base: The system voltampere base in MVA.

Voltage Base: The line-line voltage base in KV.

Impedance Base: The impedance base in Ohms. This value is automatically computed when the power base and the voltage base are entered or modified.

Admittance Base: The admittance base in Siemens. It is also automatically computed as the inverse of the impedance base.

Final Results

When all the input data is entered, the results automatically will be displayed. The values for R , X , B and G are shown in three different sections, each section corresponding to Distributed, Lumped or Total, and Per Unit values, respectively. The Power Surge Impedance Loading is calculated only for the lumped section.

Intermediate Results

The intermediate values calculated to compute the R , X , B , and G values are displayed here. The computed geometric mean radius and geometric mean distance are shown in the Distributed

values section. The characteristic impedance and propagation factor are displayed in the Lumped values section.

Note: To see the specific calculations used in this program, see the Calculations section, at the end of this document.

Ampere to MVA Limit Conversion

This section converts the limits of the transmission line from Amperes to MVAs, given the voltage base, and vice versa.

Conductor Type

This section is used to add, remove, rename, and edit the information related to the conductor types. This can be done in two ways: using the form for an individual conductor type, or using the table for all the conductor types available.

Edit By Form

Conductors are identified by a unique code word. All the available conductors are listed in the Conductor Code Word combo box. By selecting one conductor, all its properties are displayed in the form. There, the user is able to modify any of those values. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new conductor will be shown. By clicking on **Rename**, a new name for the current conductor type is required. In order to save the current conductor type with a different name is necessary to click on **Save As**. Finally, to remove the current conductor the user can do so by clicking on the **Delete** button.

Edit By Table

In this tab, all the conductor types are shown as records in a table, where every field is a characteristic of the conductor. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

Conductor Properties

The available properties of the conductors are the following:

Code Word: Code name for the type of conductor. The names of bird species are typically used. Code Word has to be unique.

Area: The area of aluminum conductor in circular mils. A circular mil is the cross-sectional area of a circular conductor having a diameter of 1 mil. One mil is one thousandth of an inch (0.001").

Aluminum strands: Number of aluminum strands.

Steel layers: Number of steel strands.

Aluminum layers: Number of aluminum layers.

External diameter: Outside diameter of the conductor in inches.

GMR: Geometric Mean Radius in feet.

DC Resistance: DC resistance of the conductor at 20°C per 1 mile in Ohms.

AC Resistance 25: AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms.

AC Resistance 50: AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms.

AC Resistance 75: AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms.
Inductive Reactance: Inductive reactance per conductor at 1 foot spacing at 60 Hz in Ohms/mile.
Capacitive Reactance: Capacitive reactance per conductor at 1 foot spacing at 60 Hz in MegaOhms/mile.

Tower Configuration

This part is used to add, remove, rename, and edit the information related to the tower configurations. This can be done in two ways: using the form for an individual tower configuration, or using the table for all the tower configurations available.

Edit By Form

Tower configurations are identified by a unique name. All the available tower configurations are listed in the Tower Configuration Name combo box. By selecting one specific tower configuration, all its characteristics are displayed in the form. There, the user can modify any of those characteristics. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new tower configuration will be shown. By clicking on **Rename**, a new name for the current tower configuration is required. In order to save the current tower configuration with a different name is necessary to click on **Save As**. Finally, to remove the current tower configuration the user can do so by clicking on the **Delete** button.

Edit By Table

In this tab, all the tower configurations are shown as records in a table, where every field is a value of the tower configuration. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

Tower Configuration Values

The values of the tower configuration are the following:

Name: Name for the tower configuration. Name has to be unique.

Phase spacing: x and y coordinates of phases A, B and C positions. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the phase spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Conductors per bundle: This section specifies the number of conductors (either single conductor or a bundle of conductors) per phase. The maximum number of conductors per bundle allowed is 4.

Use regular spacing: When using a bundle of conductors, checking the **Use Regular Spacing of** check box will use the specified value as a regular spacing among the conductors. If the **Use Regular Spacing of** check box is not checked, then the custom conductors spacing section has to be used.

Conductors spacing: x and y coordinates of the conductors in the bundle. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the bundle spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Temperature: Assumed temperature in Fahrenheit degrees for English system and Celsius degrees for Metric (SI) system.

Frequency: Frequency of the system in Hertz.

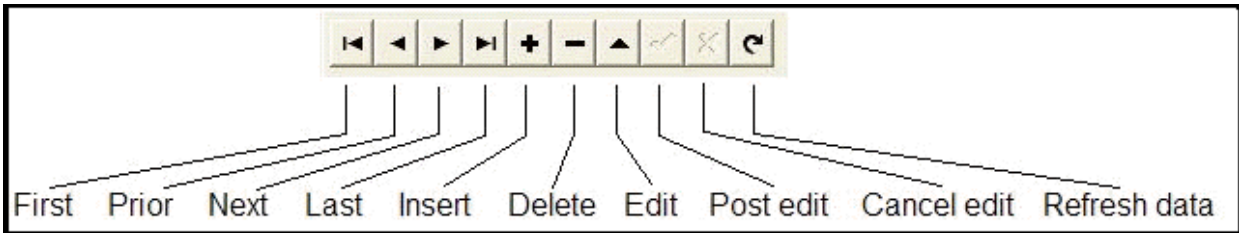
System of units: The system of units used to specify the values of the tower configuration. The options are English system or Metric (SI) system. The units specified here not necessarily have to match the units specified in the **Input Data** section.

Database

The conductor type and tower configurations tables are read by default from the file **conductors.mdb**, which is a MS Access® database. This database can be read from another *.mdb file by clicking on the **Select Conductors and Configurations Database** button.

Note: The conductors.mdb file can also be viewed and edited in MS Access®.

In order to edit a record in the database tables, the user can use the toolbar designed to do that. Following there is a figure showing this toolbar:



The **First**, **Prior**, **Next**, and **Last** buttons are used to move among records. The **Insert**, **Delete** and **Edit** buttons are used to insert, delete or edit the current record, respectively. While editing a record, the user can not change of tab until modifications are posted through the **Post edit** button or canceled with the **Cancel edit** button. The **Refresh data** button just refreshed the data of the entire table.

Calculations

Distributed Parameters

Resistance

$$R_t = \frac{\left(R_{25} + \left(\frac{R_{25} - R_{50}}{25 - 50} \right) (t - 25) \right)}{N} \quad 0 \leq t \leq 50$$

$$R_t = \frac{\left(R_{50} + \left(\frac{R_{50} - R_{75}}{50 - 75} \right) (t - 50) \right)}{N} \quad 50 \leq t$$

Where:

R_t AC resistance at temperature t per phase per 1 mile in Ohms

t Assumed temperature in Celsius degrees

R_{25} AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms

R_{50} AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms

R_{75} AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms

N Number of conductors per phase

Inductive Reactance

$$X_L = 4\pi f \times 10^{-7} \ln \frac{D_{eq}}{D_{sL}}$$

Where:

X_L Inductive reactance in Ohms/meter

f Frequency of the system in Hertz

D_{eq} Geometric mean distance between phases in meters

D_{sL} Geometric mean radius between conductors of one phase in meters

The **geometric mean distance** between phases is defined as:

$$D_{eq} = \sqrt[3]{d_{ab} d_{bc} d_{ca}}$$

Where:

d_{ab}, d_{bc}, d_{ca} Distances between phases a-b, b-c, c-a, respectively in meters

The **geometric mean radius** between conductors of one phase is defined as:

$$D_{sL} = GMR \quad \text{For 1 stranded conductor}$$

$$D_{sL} = e^{-1/4} r \quad \text{For 1 solid cylindrical conductor}$$

$$D_{sL} = \left(\prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

D_{sL} Geometric mean radius in meters

r External radius of conductor in meters

GMR Geometric mean radius given in tables for one stranded conductor

d_{km} Distance between conductors k and m in meters.

Note: If $k = m$, then $d_{km} = D_{sl}$ for one stranded or solid cylindrical conductor.

Susceptance

$$B = 2\pi f \left(\frac{2\pi\epsilon}{\ln \left(\frac{D_{eq}}{D_{sc}} \right)} \right)$$

Where:

B Susceptance in Siemens/meter

f Frequency of the system in Hertz

ϵ Constant permittivity = 8.85418×10^{-12}

D_{eq} Geometric mean distance between phases, defined as above

D_{sc} Geometric mean radius between conductors of one phase using external radius in meters

The geometric mean radius between conductors of one phase using external radius is defined as:

$$D_{sc} = r \quad \text{For 1 conductor}$$

$$D_{sc} = \left(\prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

D_{sc} Geometric mean radius in meters

r External radius of conductor in meters

d_{km} Distance between conductors k and m in meters.

Note: If $k = m$, then $d_{km} = r$.

Conductance

Assumed $G = 0$

Where:

G Conductance in Siemens/meter

Lumped (Total) Parameters

Resistance, Inductive Reactance, Conductance and Susceptance, using the equivalent π circuit (long line)

$$Z' = R' + jX' = Z_c \sinh \gamma \ell$$

$$Y' = G' + jB' = \frac{2}{Z_c} \tanh \frac{\gamma \ell}{2}$$

Where:

Z' Total series impedance of line in Ohms

Y' Total series admittance of line in Siemens

R' Total series resistance of line in Ohms

X' Total series inductive reactance of line in Ohms

G' Total series conductance of line in Siemens

B' Total series susceptance of line in Siemens

Z_c Characteristic impedance in Ohms

γ Propagation constant in meters⁻¹

ℓ Line length in meters

The characteristic impedance and propagation constant are defined as:

$$Z_c = \sqrt{\frac{z}{y}}$$
$$\gamma = \sqrt{zy'}$$

Where:

z Distributed series impedance in Ohms/meter

y Distributed series admittance in Siemens/meter

The **distributed series impedance** and **distributed series admittance** are defined as:

$$z = R + jX$$

$$y = G + jB$$

Where:

R Distributed series resistance in Ohms/meter

X Distributed series inductive reactance in Ohms/meter

G Distributed series conductance in Siemens/meter

B Distributed series susceptance in Siemens/meter

Surge Impedance Loading

The surge impedance loading is defined as the power delivered by a lossless line to a load resistance equal to the surge (or characteristic) impedance Z_0 and is given by:

$$P_{SIL} = \frac{|V_N|^2}{Z_0}$$

Where:

P_{SIL} Total surge impedance loading in a three-phase line in VA

V_N Line-line nominal voltage in Volts

Base Values

Impedance Base

$$Z_B = \frac{(V_B^N)^2}{S_B^{3\phi}}$$

Where:

Z_B Impedance base in Ohms

$S_B^{3\phi}$ Power base in VA

V_B^N Line-line voltage base in Volts

Admittance Base

$$Y_B = \frac{1}{Z_B}$$

Where:

Y_B Admittance base in Siemens
 Z_B Impedance base in Ohms

Per Unit (PU) Parameters

Resistance, Inductive Reactance, Conductance, Susceptance

$$R_{PU} = \frac{R'}{Z_B}$$

$$X_{PU} = \frac{X'}{Z_B}$$

$$G_{PU} = \frac{G'}{Y_B}$$

$$B_{PU} = \frac{B'}{Y_B}$$

Where:

R_{PU} Per unit resistance
 R' Total series resistance in Ohms
 X_{PU} Per unit Inductive reactance
 X' Total series inductive reactance in Ohms
 G_{PU} Per unit conductance
 G' Total series conductance in Siemens
 B_{PU} Per unit susceptance
 B' Total series susceptance in Siemens
 Z_B Impedance base in Ohms
 Y_B Admittance base in Siemens

MVA To Ampere and Ampere To MVA Limits Conversion

MVA to Ampere Limit Conversion

$$Lim_{Amp} = \frac{Lim_{MVA} \times 10^6}{\sqrt{3} |V_N|}$$

Where:

Lim_{Amp} Limit in Amperes

Lim_{MVA} Limit in MVAs

V_N Nominal voltage in Volts

Ampere to MVA Limit Conversion

$$Lim_{MVA} = Lim_{Amp} \frac{\sqrt{3} |V_N|}{10^6}$$

Where:

Lim_{Amp} Limit in Amperes

Lim_{MVA} Limit in MVAs

V_N Nominal voltage in Volts

Branch Options: Transformer Control

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

Options for specifying transformer control are found on the **Transformer Control** tab of the [Branch Options dialog](#) when in Edit mode and the **Transformer** tab of the [Branch Information Dialog](#) when in Run mode.

Off-nominal Turns Ratio and Phase Shift Degrees

The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40° . The phase angle field can be non-zero for LTC and fixed transformers, most notably $\pm 30^\circ$ if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

Automatic Control Type provides the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met:

- For an LTC transformer, the transformer's **Automatic Control Enabled** field must be checked. This field can also be modified on the [Transformer Records display](#). When the transformer is a phase shifter, the Automatic Control Enabled field has three options. Select *Not Enabled* to completely disable the phase shifter for automatic control. Select *Enabled for Power Flow* to allow automatic control actions during the power flow solution. Select *Enabled for OPF Only* to allow the phase shifter to be a control as part of the OPF solution but not on automatic control as part of the power flow solution.
- The transformer's area must have automatic transformer control enabled. This is specified on the **Options Tab** of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the **Power Flow Solution Tab** of the [Simulator Options Dialog](#). Automatic control for LTC transformers and phase shifters can be disabled separately at the case level.

The type of transformer can be changed on the Transformer Control Info dialog opened by clicking the **Change Automatic Control Options...** button. This dialog contains different information depending on the type of transformer control that is selected. The **Common Options** tab of this

dialog allows changing the control type while in Edit mode and displays the type of control while in Run mode. The type CANNOT be modified while in Run mode. The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

The type-specific information contained on the **Common Options** tab is described below:

No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). The Transformer Control Info dialog is configured as the [Transformer AVR Dialog](#) when this control type is selected. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution Advanced Options](#).

Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. When this control type is selected, the Transformer Control Info dialog is configured as the [Transformer Mvar Control Dialog](#).

Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). When this control type is selected, the Transformer Control Info dialog is configured as the [Transformer Phase Shifting Dialog](#). The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase AngleStep Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

Area and Case Control Options

This set of options is available when in Run mode and provides a summary of the area and case level options for allowing automatic transformer control. These settings are provided for informational purposes only and cannot be set here.

Area Transformer Control Enabled has to be checked for any transformer in that area to be controlled (phase shifter control must be enabled for power flow); **Case LTC Transformer Control Enabled** has to be checked for any LTC type (Voltage Regulation (AVR) or Reactive Power Control) transformer in the entire case to be controlled, and **Case Phase Shifter Control Enabled** has to be checked for any phase shifter in the entire case to be controlled.

Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [Power Flow Solution General Options](#).

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this dialog, click the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Voltage Regulation (AVR)* option is chosen from the **Automatic Control Type** group.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the **Regulation Minimum Voltage** and **Regulation Maximum Voltage** fields), the **Voltage Error** field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on **Power Flow Solution Tab** of the [Simulator Options dialog](#).

Line Drop/Reactive Current Compensation

Typical specification of the bus at which the voltage should be controlled by a transformer is an actual bus in the power system model. However, some control schemes require that the voltage at some fictitious bus a specified impedance away from the transformer be controlled instead. This can be accomplished using the line drop and reactive current compensation control method.

This method requires that the regulated bus for a transformer be specified as one of the transformer's terminal buses. This provides a reference point for specifying the location of the fictitious bus that is the actual controlled point. An impedance must also be specified that will determine the location of the fictitious bus in reference to the selected regulated bus.

Check the **Use Line Drop/Reactive Current Compensation** checkbox to use this control method. Specify the impedance for the location of the fictitious controlled bus using the **Line Drop/Reactive Current Compensation Resistance (R)** and **Line Drop/Reactive Current Compensation Reactance (X)** fields. The impedance should be specified in per-unit.

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Branch Options: Series Capacitor

The Series Capacitor tab of the [Branch Options](#) dialog displays information related to a series capacitor, including its status.

Status

The capacitor itself has two status positions, **Bypassed** and **In Service**. When the series capacitor is in service, the branch is modeled as a reactive branch, using the line parameters from the Parameters page. If the capacitor is bypassed, a low impedance branch is introduced to bypass the capacitor. Note that this is not the same as the branch status of **Open** or **Closed**. The branch status is the indicator of whether or not the entire circuit is operating (closed), regardless of **Bypassed** or **In Service** status on the capacitor itself.

Is Series Capacitor

If this box is checked, the branch can be treated as a series capacitor, with the series Status of Bypassed or In-Service available.

Multi-Section Line Information

Multi-section line records are used to group a number of series-connected transmission lines together so that their status is controlled as a single entity. They are usually used to model very long transmission lines that require multiple individual transmission line records if they are to be modeled accurately. In terms of line status, Simulator then treats each multi-section line as a single device. In other words, changing the status of one line in the record changes the status of the other lines in the record, as well.

Multi-section line records are **not** directly represented on the oneline diagrams. Rather, to view this dialog, right-click the record of interest on the [Multi-Section Line Records](#) display and select **Show Dialog** from the resulting local menu. To define a new multi-section line, switch to Edit Mode and, from the Multi-Section Line Records display, right-click and select **Insert** from the resulting local menu.

Note: If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well.

The Multi-Section Lines Dialog contains the following fields:

From Bus Number and Name

Number and name of the from bus for the record. If you are defining a new multi-section line, this must be the first data item you specify.

To Bus Number and Name

Number and name of the "to" bus for the record. You cannot specify this value directly. Instead, use the table at the bottom of the dialog to define the intermediate buses and *to* terminal.

Circuit

Two character identifier used to distinguish between multiple records joining the same from/to buses. The first character of the circuit identifier must be an "&."

You can use the spin button immediately to the right of the circuit field to view other multi-section line records. However if you have changed the record, **you must** select **Save** before moving to another record. Otherwise, your changes will be lost.

From End Metered

If checked, the *from end* of the record is the metered end; otherwise the *to end* is metered.

Multi-Section Line Name

The name for the multi-section line.

Table

The table lists the dummy (or intermediate) buses and the *to bus* that comprise the record. The first column should contain the first dummy bus number and the circuit ID of the transmission line joining the *from bus* with the first dummy bus. The next column should contain the second dummy bus number and the circuit id of the transmission line joining this bus to the first dummy bus. Continue until the last column contains the *to* bus and the circuit ID of the transmission line joining the last dummy bus with the "to" bus.

Each transmission line comprising the multi-section line must already exist. If only one transmission line joins any two buses, you may omit the circuit identifier.

OK

Saves any modifications and closes the dialog.

Save

Saves any modifications but does not close the dialog.

Cancel

Closes the dialog without saving modifications to the current record.

Delete

Deletes the current multi-section line record.

Note: If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well. One exception is if a section or sections of a multi-section line are removed by [merging](#) the section terminal buses together into a single bus. If special conditions are met, the multi-section line record will be maintained.

Line Field Options

Line field objects are used to show different values associated with transmission lines, transformers and DC lines. This dialog is used to view and modify the parameters associated with these fields.

Find...

If you do not know the exact line identifiers you are looking for, you can click this button to open the [advanced search engine](#).

Near Bus Number

Bus associated with the near end of the object. All fields display values calculated at the *near bus* end. When inserting fields graphically, this field is automatically set to the closest bus on the oneline.

Far Bus Number

Bus associated with the *far end* of the object.

Circuit

Two-character identifier used to distinguish between multiple lines or transformers joining the same two buses. Default is '1'.

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta Per Mouse Click

Only used with the MVA Limit field type. Specifying a nonzero value for this field equips the MVA Limit Field with an integrated spin button that can be clicked to increment or decrement the MVA Limit by the amount of the *Delta Per Mouse Click* value.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Anchored

If checked, the line analog is [anchored](#) to its associated line.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of line field to show. The following choices are available:

AC Line MW Flow MW flow into the line or transformer at the near bus

AC Line Mvar Flow Mvar flow into the line or transformer at the near bus

AC Line MVA Flow Magnitude of MVA flow into the line or transformer at the near bus

AC Line Amp Flow Magnitude of Amp flow (in amps) into the line at the near bus

AC Line MW Losses Real power losses on the line or transformer in MW

AC Line Mvar Losses Reactive power losses on the line or transformer in Mvar

DC Line MW Flow MW flow into the DC line at the near bus

DC Line Mvar Flow Mvar flow into the DC line at the near bus

MVA Limit MVA limit for the line or transformer

DC Line Set point Set point value for the DC line; see [DC Transmission Line dialog](#)

Select a Field Choose from over 80 different fields

Select **OK** to save changes and close the dialog or Cancel to close the dialog without saving your changes.

Series Capacitor Field Options Dialog

Series capacitor field objects are used to show field values specific to series capacitors. Use [Line Fields](#) to show fields generic to transformers, series capacitors and transmission lines, such as the flow of power through the device. The series capacitor fields dialog is used to view and modify the parameters associated with series capacitor specific fields.

Near Bus Number

Bus associated with the *near end* of the series capacitor.

Far Bus Number

Bus associated with the *far end* of the series capacitor.

Circuit

Two-character identifier used to distinguish between branches joining the same two buses. Default is '1'.

Anchored

When checked, the text field will move with the series capacitor if the series capacitor is moved on the oneline diagram.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text will be placed.

Field Value

The value of the currently selected field.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Status Capacitor status

Series Capacitance Capacitance of series capacitor

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

Transformer Field Options Dialog

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

Near Bus Number

Bus associated with the *near end* of the transformer.

Far Bus Number

Bus associated with the *far end* of the transformer.

Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Off-nominal Tap Ratio Actual tap ratio

Phase Shift Angle Actual phase shift in degrees

Off-nominal Tap Position Tap position in steps, usually ranging from L16 to R16

Automatic Control Status The status of the control for the transformer

Select **OK** to save changes and close the dialog or Cancel to close dialog without saving your changes.

DC Transmission Line Options

This dialog is used to view and modify the parameters associated with each two-terminal DC transmission line in the system. It can also be used to insert new two-terminal DC transmission lines or to delete existing ones.

This dialog has separate pages: [Line Parameters](#), [Rectifier Parameters](#), [Inverter Parameters](#), [Actual Flows](#), [OPF](#), [Custom](#), and [Stability](#). The Actual Flows page is not used in Edit Mode. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages can be used to view/change the modeling parameters associated with the DC lines.

DC Line Options: Line Parameters

This page is used to enter parameters associated with the DC line itself.

Rectifier and Inverter Bus Numbers and Names

These fields indicate the numbers and names of the rectifier and inverter ends of the line. When graphically inserting a DC line, these fields are automatically determined based upon the starting and ending buses used in drawing the line. When investigating existing DC line records, you may use the *Find By Numbers* button to identify a DC line between a specific rectifier - inverter pair. You may also use the spin control to cycle through the list of DC line records modeled in the case.

Circuit ID

This field is not currently used, since only one DC transmission line can exist between each inverter-rectifier pair. It is hard-coded to be 1.

Find By Numbers

To find a DC line by its bus numbers, enter the *Rectifier* and *Inverter* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of DC lines in the system.

Area Name

Names of the areas in which the rectifier and inverter buses are located.

Labels

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the DC line. New labels can also be added for the DC Line from the dialog as well.

Link to New DC Line

Use the *Link to New DC Line* button to create a new DC line corresponding to the entries you have made in the dialog. This button performs the same function as pressing *Save*. Note that adding a new DC line to the case in this way does not add a DC line to the online display; the new DC line is present only in the model. You may then add the newly modeled DC line to the online diagram in the usual way (such as select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

Line Parameters

Status

Operating status of the DC transmission line. If the status is Open, then no power can flow on the DC line regardless of the control mode setting. If the status is closed, then the control mode will dictate how power flows on the line.

Control Mode

The initial control mode for the line. Specify *Blocked* to disable the DC line, *Power* to maintain specified MW power flow through the line, or *Current* to maintain specified current flow through the line.

Setpoint

If the line operates in the Power Control Mode, *Setpoint* should indicate the desired power flow in MW. To specify power flow at the rectifier end, enter a positive value. Enter a negative value to

specify power flow at the inverter end. If the DC line operates in Current Control Mode, enter the desired line flow in amps.

Resistance

Resistance of the DC Line.

Sched. Voltage

Scheduled DC line voltage in kV. The value of Rcomp is used to determine whether this value specifies the inverter end or the rectifier end.

Switch Voltage

When the line operates in the Power Control Mode, this is the inverter voltage level in kV at which the line switches from constant power to constant current control.

RComp

Compounding resistance. The compounding resistance dictates whether the inverter voltage is scheduled ($R_{comp} = 0$), or the rectifier voltage is scheduled ($R_{comp} = \text{DC line resistance}$). Simulator does not currently support scheduling the voltage at a point along the DC line ($0 < R_{com} < \text{DC line resistance}$), but instead will treat any non-zero value of Rcomp as scheduling the voltage at the rectifier.

Setpoint Specified At

Indicates which end of the DC transmission line the Setpoint value is designated. This will be the terminal where the setpoint value is maintained. The opposite terminal flow value will be a calculated quantity.

Metered End of Line

Indicates which end of the DC line is assumed metered for Area interchange calculations.

DC Line Options: Rectifier Parameters

This page is used to enter parameters associated with the rectifier end of the line.

of Bridges

Number of valve bridges in series.

Base Voltage

Base AC voltage in kV on primary side of transformer.

XF Ratio

Transformer ratio.

XF Tap

Transformer tap setting.

XF Min/Max Tap, XF Tap Step

Transformer minimum and maximum tap settings, and the tap's step size.

Commuting XF Resistance and Reactance

Commuting resistance and reactance for the transformer, in ohms.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

DC Line Options: Inverter Parameters

This page is used to enter parameters associated with the inverter end of the line. Entries are identical to the Rectifier Page, except here they are associated with the inverter.

of Bridges

Number of valve bridges in series.

Base Voltage

Base AC voltage in kV on primary side of transformer.

XF Ratio

Transformer ratio.

XF Tap

Transformer tap setting.

XF Min/Max Tap, XF Tap Step

Transformer minimum and maximum tap settings, and the tap's step size.

Commuting XF Resistance and Reactance

Commuting resistance and reactance for the transformer, in ohms.

Minimum, Maximum, and Actual Firing Angle

Minimum, maximum, and actual values of the firing angle for the rectifier.

DC Line Options: Actual Flow

Shows the actual real and reactive power flows into the line at the rectifier and the inverter, as well as the voltage at both ends (in kV) and the line current in amps. The contents of this page are valid only when Simulator is in Run Mode.

DC Line Options: OPF

This page is used to enter parameters associated with the OPF.

Rectifier/Inverter Bus MW Marginal Costs

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

OPF Control**Minimum and Maximum Setpoint Value**

The minimum and maximum setpoint allowed for the DC Line. When on OPF Control the DC Line will be dispatched between these values.

Current Setpoint Value

The current Setpoint as found on the Line Parameters Tab.

Transmission Charge

This is the cost incurred for moving power across the DC transmission line. The cost is zero at a zero MW flow and increases linearly using this charge.

OPF Control Enabled for this DC line

This check box must be checked if the DC limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the DC line to violate its limits.

Include Impact of DC Line Marginal Losses

When dispatching the DC Line this determines whether the marginal losses from the DC line are taken into account.

VSC DC Line Dialog

Simulator allows the modeling of newer types of Voltage Source Converter DC lines, which use Insulated Gate Bipolar Transistors (IGBT) and Gate Turn-off Thyristors (GTO) to perform pulse-width modulation or multi-level switching. They are also referred to as "HVDC Light" (ABB) or "HVDC Plus" (Siemens).

For power flow purposes, these devices are simple: they completely decouple real and reactive power controls.

There is currently no way to display VSC DC lines on oneline diagrams. However, they are displayed in [Bus View](#) (in a similar manner to [Multi-Terminal DC Lines](#)), [Power Flow](#), and [Quick Power Flow](#) lists.

VSC DC Name

Name of the VSC DC Transmission Line. This is the unique identifier for the device. The dialog field is not editable itself, but the name can be changed with the "Rename" button.

To/From Converter

Gives information about the endpoint buses: the numbers, names, and associated Substations and Areas. These fields are not editable, the information displayed here can only be changed through the buses themselves. The connected buses can be changed on the **Converter Parameters** tab below.

Status

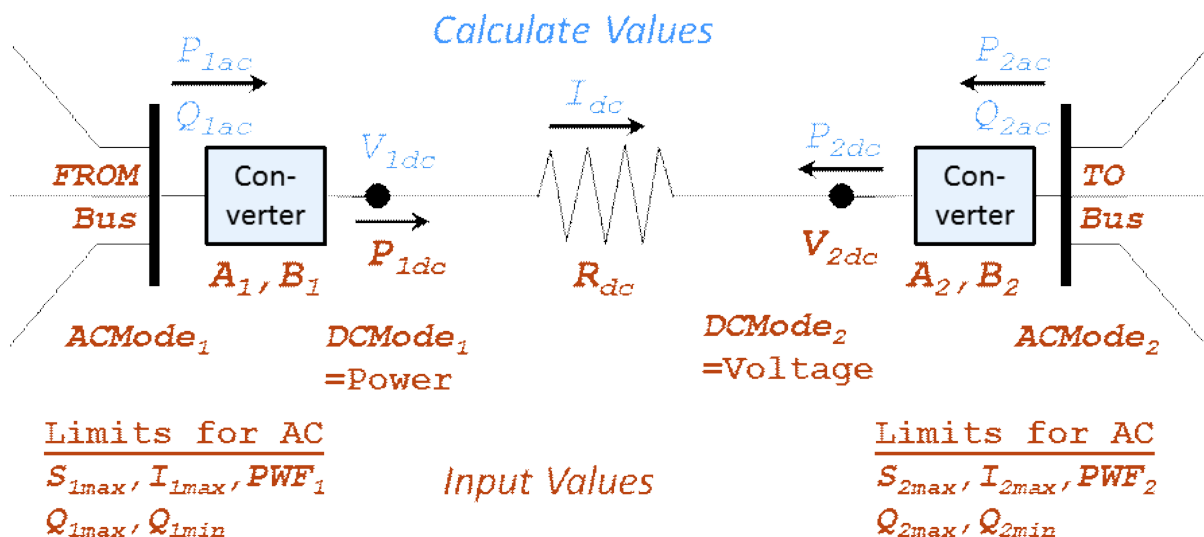
Allows the line to be opened or closed.

Resistance

Set the resistance of the line.

Converter Parameters Tab

This tab gives control over the parameters for the converters on each end of the line.



Terminal Bus

Allows the connected buses to be set.

DC Setpoint

Specifies the DC control mode for each converter, either Out-Of-Service, Voltage, or Power.

	To DC Mode = <i>Out</i>	To DC Mode = <i>Voltage</i>	To DC Mode = <i>Power</i>
From DC Mode = <i>Out</i>	DC Line is ignored	Treat as SVC at To Bus if AC Mode = <i>Voltage</i>	
From DC Mode = <i>Voltage</i>	Treat as SVC at From Bus if AC Mode = <i>Voltage</i>	DC Line is ignored	Valid two-terminal region
From DC Mode = <i>Power</i>		Valid two-terminal region	DC Line is ignored

AC Setpoint

Specifies the AC control mode for each converter, either per unit Voltage or Power Factor. If the AC control mode is set to Voltage and the other converter is out of service, it acts as an SVC.

Regulated Bus, Remote Regulation %

When the AC mode is Voltage, the Regulated Bus does not have to be the converter AC terminal. If this is blank, it signifies it regulated its own terminal. For more information on Remote Regulation, refer to the [Generator Information Power and Voltage Control page](#).

A loss, B loss

The converter losses for each end are defined as:

$$\text{ConverterLoss kW} = A_{\text{loss}} + I_{\text{DC}} * B_{\text{loss}}$$

Minimum Loss

If this field is specified, it sets a lower bound on the value calculated from A_{loss} , B_{loss} and I_{DC}

Maximum AC MVA

Converter MVA rating specified in MVA. If set to 0, there is no limit.

Maximum AC Current

Converter current rating in amps. If set to 0, there is no limit.

Power Weighting Factor

Power weighting factor is a value between 0 and 1. It is used to reduce real and reactive power when limits are hit.

Maximum Mvar

Maximum Mvar injection into the AC network. This is used only when the AC mode is Voltage.

Minimum Mvar

Minimum Mvar injection into the AC network. This is used only when the AC mode is Voltage.

Flows and Voltages Tab

These fields are not editable.

AC MW Flow

The real power flow on the AC side of the converter.

AC MVar Flow

The reactive power on the AC side of the converter.

DC MW Flow

The power flow on the DC side of the converter.

DC Voltage

The voltage on the DC side of the converter, in kV.

DC Line Current

The current flowing on the DC line, in amps.

Custom Tab

Tab containing [Custom Fields](#).

Stability Tab

Tab for configuring the VSCDC Line Model for stability analysis

Multi-Terminal DC Record Information

This dialog is used to view and modify the parameters associated with multi-terminal DC records. It is also used when inserting new multi-terminal DC records.

The Multi-Terminal DC Record dialog can be used to inspect and modify the model of a multi-terminal DC network record. To view the Multi-Terminal DC Record, simply right-click on the record of interest in the [Multi-Terminal DC Record Display](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

Record Number

Unique number between 1 and 999 which identifies the current multi-terminal DC record.

Number of Devices

Lists the number of DC buses, converters, and DC Lines that form the multi-terminal DC network.

Control

The control method used when solving the multi-terminal DC network.

Controlling Converter

The AC converter bus number where the DC voltage is being controlled.

MTDC Network Status

Status of the entire Multi-terminal DC network. If this field is set to Closed, the entire DC subnetwork of the MTDC model is considered disconnected in the load flow case.

DC Buses Tab

This page of the dialog displays the DC bus records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC bus, right-click on the record of interest and choose *Show Dialog* from the popup menu.

DC Converters Tab

This page of the dialog displays the DC converter records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC converter, right-click on the record of interest and choose *Show Dialog* from the popup menu.

DC Lines Tab

This page of the dialog displays the DC line records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC line, right-click on the record of interest and choose *Show Dialog* from the popup menu.

Custom Tab

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom strings and values can also be entered and saved with the case file as well.

Multi-Terminal DC Bus Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

DC Bus Number, DC Bus Name

The number and name identifiers for the selected DC bus.

AC Bus Number

The AC bus number connected to the DC bus through an AC/DC converter. If the selected DC bus is a bus that is internal to the DC multi-terminal network (not directly connected to an AC bus,) this field will be 0.

Area Number, Area Name

The number and name identifiers of the control area the DC bus is contained in.

Zone Number, Zone Name

The number and name identifiers of the zone the DC bus is contained in.

Ground Resistance

Resistance to ground of the DC bus, entered in Ohms. This field is currently only for storage of values supported by other load flow formats, and is currently not used by Simulator.

Multi-Terminal DC Converter Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

Converter Parameters

Number of Bridges

Number of bridges in series for the selected converter.

Converter Type

R for rectifier or **I** for inverter.

Commutating Impedance

Commutating impedance per bridge, in Ohms.

Firing Angle Limits

The maximum and minimum firing angle limits, in degrees.

Transformer Parameters

AC Base

The primary AC base voltage, in kV.

DC Base

The DC base voltage, in kV.

Transformer Ratio

Actual transformer ratio.

Tap Settings

Displays the actual tap setting, the tap step, and maximum and minimum tap values for the converter transformer.

Control Parameters

Setpoint

The setpoint control value at the converter. For the voltage-controlling converter, this field is set to 0. For the remaining converters, this field displays MW when in Power mode, or Amps when in Current mode.

Margin

Rectifier margin, entered in per-unit of the DC power or current. This field is currently only for support of other load flow formats, and is not used by Simulator.

DC Participation Factor

Converter participation factor. This field is currently only for support of other load flow formats, and is not used by Simulator.

Voltage

The DC Voltage magnitude at the DC side of the converter.

Solved Parameters

Firing Angle

The firing angle of the converter, as determined during the load flow solution.

DC Current

The calculated DC current at the converter DC terminal.

MW, MVAR

The real and reactive power delivered to (or absorbed from) the AC system by the converter.

Multi-Terminal DC Line Information

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

From and To DC Bus Number

The DC bus numbers of the From and To buses in the multi-terminal DC network. These fields must contain valid DC bus numbers of the selected multi-terminal DC record. AC bus numbers from the load flow case are not acceptable bus numbers for a multi-terminal DC line.

DC Circuit

The circuit identifier for the DC line.

DC Resitance and DC Inductance

The resistance and inductance of the DC line. Resistance is in Ohms, and is used for solving the load flow of the DC network. The inductance is in milliHenries, and is not used for solving the load flow. The inductance field is currently only for support of other load flow formats, and is not used by Simulator.

Branch Options: Transformer Control

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

Options for specifying transformer control are found on the **Transformer Control** tab of the [Branch Options dialog](#) when in Edit mode and the **Transformer** tab of the [Branch Information Dialog](#) when in Run mode.

Off-nominal Turns Ratio and Phase Shift Degrees

The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40° . The phase angle field can be non-zero for LTC and fixed transformers, most notably $\pm 30^\circ$ if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

Automatic Control Type provides the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met:

- For an LTC transformer, the transformer's **Automatic Control Enabled** field must be checked. This field can also be modified on the [Transformer Records display](#). When the transformer is a phase shifter, the Automatic Control Enabled field has three options. Select *Not Enabled* to completely disable the phase shifter for automatic control. Select *Enabled for Power Flow* to allow automatic control actions during the power flow solution. Select *Enabled for OPF Only* to allow the phase shifter to be a control as part of the OPF solution but not on automatic control as part of the power flow solution.
- The transformer's area must have automatic transformer control enabled. This is specified on the **Options Tab** of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the **Power Flow Solution Tab** of the [Simulator Options Dialog](#). Automatic control for LTC transformers and phase shifters can be disabled separately at the case level.

The type of transformer can be changed on the Transformer Control Info dialog opened by clicking the **Change Automatic Control Options...** button. This dialog contains different information depending on the type of transformer control that is selected. The **Common Options** tab of this

dialog allows changing the control type while in Edit mode and displays the type of control while in Run mode. The type CANNOT be modified while in Run mode. The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

The type-specific information contained on the **Common Options** tab is described below:

No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). The Transformer Control Info dialog is configured as the [Transformer AVR Dialog](#) when this control type is selected. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution Advanced Options](#).

Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. When this control type is selected, the Transformer Control Info dialog is configured as the [Transformer Mvar Control Dialog](#).

Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). When this control type is selected, the Transformer Control Info dialog is configured as the [Transformer Phase Shifting Dialog](#). The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase AngleStep Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

Area and Case Control Options

This set of options is available when in Run mode and provides a summary of the area and case level options for allowing automatic transformer control. These settings are provided for informational purposes only and cannot be set here.

Area Transformer Control Enabled has to be checked for any transformer in that area to be controlled (phase shifter control must be enabled for power flow); **Case LTC Transformer Control Enabled** has to be checked for any LTC type (Voltage Regulation (AVR) or Reactive Power Control) transformer in the entire case to be controlled, and **Case Phase Shifter Control Enabled** has to be checked for any phase shifter in the entire case to be controlled.

Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [Power Flow Solution General Options](#).

Transformers Bases and Impedances Dialog

Typically the impedances and tap values of transformers is already assumed to have been converted to unity tap base and bus nominal voltage base. However, some load flow formats provide the taps and impedances on specific transformer bases, which are different than the bus voltage and unity tap base assumptions. In these cases, Simulator will convert parameters from the transformer bases to the unity tap and bus nominal voltage base. Display of the impedances and tap values normally displayed in the [Branch Options](#) dialog are displayed on the Simulator assumed bases. However, if you wish to view the original transformer values on the transformer supplied bases, this dialog will display the original values. You can modify the original values stored here in this dialog. Note that when you do so, the converted values that Simulator stores on the system bases will also be automatically updated to reflect the change that has been made to the original values on the transformer bases.

Transformer Impedance Correction Table Display

The Transformer Impedance Correction Display shows information about all the transformer impedance correction tables in the case. The Transformer Impedance Correction Display is used to model the change in the impedance of the transformer as the off-nominal turns ratio or phase shift angle is varied.

The Correction Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated correction tables. When in Edit Mode, you can define new tables using the *Insert* option, or delete existing tables using *Delete*. You can also sort the transformer impedance correction information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Impedance Correction Tables** from the [Model Explorer](#).

The display contains the following fields by default:

Table #, Table Name

Shows the table number and name for the record. Table number must be between 1 and 64.

Transformer Impedance Scaling Factors

The rest of the default columns show the actual fields in the table. The first field of each pair shows the off-nominal turns ratio or phase shift angle, while the second field shows the associated scaling factor for the transformer's impedance.

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this dialog, click the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Voltage Regulation (AVR)* option is chosen from the **Automatic Control Type** group.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the **Regulation Minimum Voltage** and **Regulation Maximum Voltage** fields), the **Voltage Error** field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on **Power Flow Solution Tab** of the [Simulator Options dialog](#).

Line Drop/Reactive Current Compensation

Typical specification of the bus at which the voltage should be controlled by a transformer is an actual bus in the power system model. However, some control schemes require that the voltage at some fictitious bus a specified impedance away from the transformer be controlled instead. This can be accomplished using the line drop and reactive current compensation control method.

This method requires that the regulated bus for a transformer be specified as one of the transformer's terminal buses. This provides a reference point for specifying the location of the fictitious bus that is the actual controlled point. An impedance must also be specified that will determine the location of the fictitious bus in reference to the selected regulated bus.

Check the **Use Line Drop/Reactive Current Compensation** checkbox to use this control method. Specify the impedance for the location of the fictitious controlled bus using the **Line Drop/Reactive Current Compensation Resistance (R)** and **Line Drop/Reactive Current Compensation Reactance (X)** fields. The impedance should be specified in per-unit.

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Mvar Control Dialog

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this dialog, click on the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Reactive Power Control* option is chosen from the **Automatic Control Type** group.

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the **Regulation Minimum Mvar Flow** and **Regulation Maximum Mvar Flow** fields, the **Mvar Error** field indicates by how much the flow falls outside the control range.

Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from* bus. The transformer attempts to regulate the reactive flow to fall within this range.

Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

Present Tap Ratio

The transformer's present off-nominal turns ratio.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Phase Shifting Dialog

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this dialog, click on the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Phase Shift Control* option is chosen from the **Automatic Control Type** group.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the **MW Error** field indicates by how much the flow violates the regulating range.

Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, the transformer will not change its phase angle. When the regulated MW flow moves outside of this regulation range, Simulator will calculate a new phase angle in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in phase angle. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

Step Size (Degrees)

Phase shift change per step in degrees.

MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the phase angle. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the phase angle changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Field Options Dialog

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

Near Bus Number

Bus associated with the *near end* of the transformer.

Far Bus Number

Bus associated with the *far end* of the transformer.

Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Designates the type of transformer field to show. The following choices are available:

Off-nominal Tap Ratio Actual tap ratio

Phase Shift Angle Actual phase shift in degrees

Off-nominal Tap Position Tap position in steps, usually ranging from L16 to R16

Automatic Control Status The status of the control for the transformer

Select **OK** to save changes and close the dialog or Cancel to close dialog without saving your changes.

Three Winding Transformer Information

The Three Winding Transformer Dialog is used to create, modify or delete three winding transformer records in Edit Mode, or to view information for a specific three winding transformer record in run mode. Note that all the values displayed on this dialog are on the system MVA base. If the records were created from a file, the values are automatically converted to the system base. If a three winding transformer record is entered manually, the parameters need to be entered into Simulator computed with the same system MVA base Simulator is using.

This dialog has the following controls:

Primary Winding

This section of the dialog displays the primary winding terminal bus number, nominal kV, and fixed tap value (in per unit). In addition, the automatic tap changer is assumed to be on the primary winding of a three winding transformer. Therefore the LTC field displays the tap changer tap value (in per unit) on the primary winding.

Secondary Winding

This section of the dialog displays the secondary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

Tertiary Winding

This section of the dialog displays the tertiary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

Star Bus (Internal Node)

This section of the dialog displays the internal node parameters of the three winding transformer model. Three winding transformers are modeled as three two winding transformers connected at the three winding transformer terminal buses to a common or internal node, referred to as the star bus. The parameters displayed for the star bus are the bus number, voltage (in per unit), and angle.

Primary-Secondary, Secondary-Tertiary, and Tertiary-Primary Impedance

These are the actual three winding transformer winding to winding impedances, in per unit on the system base. These values are used to compute the equivalent two winding transformer impedances for the two winding transformers used to model the three winding transformer operation.

Transformer Parameters

The values for the magnetizing conductance (G) and magnetizing susceptance (G), in per unit on the system base.

Circuit ID

The circuit identifier for the three winding transformer.

Status

The status of the three winding transformer. If checked, the three winding transformer model is in service, otherwise the equivalent model is treated as out of service.

Mathematically equivalent two-winding transformers

This table displays the three two winding transformers that are mathematically equivalent representations of the three winding transformer. If you read the three winding transformer record from a file, the two winding equivalent transformers are created automatically. If you are inserting a three winding transformer manually, you can set the parameters for the primary, secondary and

tertiary windings in the fields above, then click the **Set Two-Winding Equivalent Transformers** button to have Simulator automatically create the two winding transformer records for you. You can also right-click in this table and insert, modify or delete two winding transformers manually if you already have the two winding transformer representations created.

Once you are finished with the dialog, you can click **Save** or **OK** to save any changes. If you wish to abandon any changes you have made, click **Cancel**.

Switched Shunt Information (Edit Mode)

The Switched Shunt Options dialog is used to view and modify the parameters associated with each switched shunt in the system. It can also be used to insert new switched shunts and sometimes to delete existing shunts. Multiple switched shunts are permitted at each bus, although only one is allowed to be on automatic control at each bus. Switched shunts usually consist of either capacitors to supply reactive power (in Mvar) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps.

This dialog can be accessed by right-clicking on a switched shunt display object and choosing **Switched Shunt Information Dialog** or right-clicking on a switched shunt record in the [Switched Shunt Display](#) and choosing **Show Dialog**.

The Edit mode version of this dialog is very similar in content to its [Run mode](#) counterpart.

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the switched shunt is attached. You can use the spin button immediately to the right of the number to move to the next switched shunt (click the up arrow) or the previous switched shunt (click the down arrow).

Find By Number

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click this button.

Bus Name

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters.

Find By Name

To find a switched shunt by its bus name, enter the bus name into the Bus Name field (case insensitive). Then click this button.

Shunt ID

Since multiple switched shunts are allowed on a single bus, each switched shunt has a unique Shunt ID.

Find...

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Open or closed status of the switched shunt.

Labels

Clicking on this button will open the [Label Manager Dialog](#) listing all the [labels](#) assigned for the selected switched shunt.

Area Number, Area Name

The area number and name to which the switched shunt belongs. Note that you can change the area of the switched shunt to be different than the area of the terminal bus. If you do so, you will be

prompted to confirm that you wish to place the switched shunt within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the switched shunt belongs. Note that you can change the zone of the switched shunt to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the switched shunt belongs. This is the same as that of the terminal bus.

Display**Display Size**

Size of the switched shunt.

Scale Width with Size

Automatically scales the width of the symbol when the object is resized.

Display Width

Width of the switched shunt symbol.

Pixel Thickness

Thickness of the display object in pixels.

Orientation

Specifies the direction in which to draw the object.

Anchored

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

Link to New Shunt

Adds a new record in the data or links the selected shunt display object to a different record.

OK, Save, Delete, and Cancel

OK saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional switched shunts. **Delete** deletes the current switched shunt; this option is not available when inserting objects graphically or opening the dialog by selecting a display shunt object– use the cut command instead. **Cancel** closes the dialog but does not save any changes.

Parameters**Nominal Mvar**

The Nominal Mvar field gives the amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

Nominal MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have

MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

Control Mode

Information about this option can be found in the [Switched Shunt Control](#) topic.

Control Regulation Settings

Information about these options can be found in the [Switched Shunt Control](#) topic.

Switched Shunt Blocks

Information about these settings can be found in the [Switched Shunt Control](#) topic.

Control Parameters: Advanced Options

Information about these options can be found in the [Switched Shunt Control](#) topic.

Control Parameters: Time Step Options

The [Time Step Options](#) tab contains control options specific for the [Time Step Simulation](#) tool.

Fault Parameters

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of blocks.

Custom

This page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the switched shunt. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Stability

This tab is only visible with the [Transient Stability Add-On](#) tool. Any switched shunt-specific transient stability modeling information is contained on this tab. For more information see the [Transient Stability Data: Object Dialogs](#).

Switched Shunt Field Information

Switched shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

The Switched Shunt Fields Information Dialog can be used to modify the properties of individual switched shunt fields on the oneline. The dialog displays the following fields:

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Bus Number

Number of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

Bus Name

Name of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

ID

ID of the switched shunt associated with the field.

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Delta Per Mouse Click

Switched shunt fields can be used not only to show various fields associated with switched shunt devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the switched shunt field. When the up spin button is clicked, the switched shunt field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the switched shunt field value is decreased by the same amount.

This field is only used for Switched Shunt Mvar fields. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the switched shunt field on the oneline.

Rotation Angle in Degrees

The angle at which the text is placed on the diagram, in degrees.

Anchored

If the *Anchored* checkbox is checked, the switched shunt field is [anchored](#) to its associated switched shunt, which means that it will move with the switched shunt.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of switched shunt field to show. The following choices are available:

Switched Shunt Mvar Total Mvar capacitance at the bus

Select a Field Choose from any of the different switched shunt fields

Select **OK** to save changes and to close the dialog, or click Cancel to close the dialog without saving your changes.

Zone Information (Edit Mode)

The Zone Dialog is used in the Edit Mode to view information about a zone and to move one or more buses from one zone to another. (See [Zone Information \(Run Mode\)](#) for help on the corresponding Run Mode version.) To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Then, right-click on the desired zone record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

Zone Number

Zone number between 1 and 999. You can use the spin button immediately to the right of this field to move to either the next zone (click the up arrow) or the previous zone (click the down arrow).

Zone Name

Alphanumeric identifier for the zone. You can use this field to change the zone's name, provided you click either **Save** or **OK**.

Find By Number

To find a zone by its number, type the number in the **Number** field and click this button.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Labels

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels assigned for the selected zone.

OK, Save, Cancel

OK saves any changes to the zone name and closes the dialog. **Save** saves any changes to the zone name, but does not close the dialog. **Cancel** closes the dialog ignoring any changes.

The zone dialog contains three additional pages of information:

Zone Buses Table

This table lists all of the buses in the zone. Number, name, voltage, area number, and area name are shown for each bus. This table can be used to move buses to a different zone. Select the bus or buses you would like to move with the mouse. Then, enter the **Destination Zone Number**, which is the zone to which you want to move the selected buses. You may enter a zone number that does not already exist, too, so that the buses will be moved to a brand new zone. In this case, be sure to provide the **Zone Name**, as well. Finally, click the **Move Selected Bus(es) to Destination Zone** button to implement the move.

OPF

The **Reserve Requirement Curves** options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Custom

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.

Bus Information (Run Mode)

This dialog is used to view information about each bus in the system. It can be displayed by right-clicking on any bus on the oneline and choosing **Bus Information Dialog** or choosing **Show Dialog** from the [Bus Display](#). This dialog can only be reached in Run Mode, but has a similar [Edit Mode](#) counterpart. The Bus Information Dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus. You can use the small arrow immediately to the right of the number to view a list of all buses in the case with valid [display filters](#). Or you can use the spin button further to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

Find By Number

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

Bus Name

Unique alphabetic identifier for the bus. You can use the small arrow immediately to the right of the bus name to view a list of all bus names in the case with [valid display filters](#).

Find By Name

To find a bus by its name, enter the bus name into the **Bus Name** field (case insensitive) and then click this button.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Nom. Voltage

Nominal voltage of the bus.

Labels

Clicking on this button will open the [Label Manager dialog](#) listing all of the labels assigned for the selected bus.

Area Number, Name

Each bus is associated with an Area record. These fields show the number and name of this area. See [Area Records Display](#) for more details about areas.

Zone Number, Name

Each bus is associated with a Zone record. These fields show the number and name of the zone. See [Zone Records Display](#) for more details about zones. You can also use the [Zone Dialog](#) to list the buses in a particular zone and to easily move a group of buses from one zone to another.

Substation Number, Name

Each bus can be associated with a Substation record. By default, buses are not assigned to substations, and in that case these fields are blank. See the [Substation Records Display](#) topic for more details on adding substations.

Owner Number, Name

Each bus is associated with an Owner record. These fields show the number and name of the owner. See [Owner Records Display](#) for more information about Owners.

Voltage (per unit)

Bus voltage in per unit notation. You may enter a new per unit voltage magnitude. However, the only effect this has is changing the initial voltage guess used in the iterative solution. If you would like to change the reference voltage for a generator, please see [Generator Information Dialog](#).

Voltage (kV)

Bus voltage in actual kilovolts.

Angle (deg) and Angle (rad)

Voltage angle at the bus in degrees and radians. You may enter a new voltage angle. However, the only effect this has is changing the initial voltage guess used in the iterative solution EXCEPT AT THE SLACK BUS. Changing the angle for the slack bus will shift the voltage angle for all the buses in the slack bus' island by a similar amount.

Status

Status of the bus, either connected or disconnected. A disconnected bus is not energized. You can use this field to change the status of the bus. When the bus is initially connected, selecting **Disconnected** opens all of the transmission lines incident to the bus, disconnecting the bus from the rest of the system. Selecting **Connected** closes all of the lines incident to the bus unless they attach to another disconnected bus.

View Substation Dialog

Clicking on this button will open the [substation dialog](#) for the substation the bus is contained in.

View Owner Dialog

Clicking on this button will open the bus' [owner dialog](#).

Slack Bus

Checked only if the bus is a system slack bus. This value can only be changed in the Edit Mode.

View All Flows at Bus

Clicking on this button will open a [quick power flow list](#) for the current bus.

Device Info

Generator Information

Displays the total MW and Mvar generation at the bus. You cannot change either of these fields from this display. Select **View/Edit Generator Records** to view the individual generator records for the bus. Selecting this button displays the [Generator Dialog](#) for the first generator at the bus.

Load Information

Displays the total MW and Mvar load at the bus. You cannot change either of these fields from this display. Select **View/Edit Bus Load Records** to view the individual load records for the bus. Selecting this button displays the [Load Dialog](#).

Shunt Admittance

Shows the real and reactive components of the shunt admittance to ground. Entered in either MW or Mvar, assuming one per unit voltage. B is positive for a capacitor and negative for a reactor. If B corresponds to a switched device, consider using a [switched shunt](#).

Bus Voltage Regulation

This section lists the devices (if any) that are controlling the voltage at the bus. Select **Bus Voltage Regulator Devices** to view the individual devices regulating the voltage for the bus. Selecting this button displays the [Bus Voltage Regulating Devices Dialog](#).

Desired PU Voltage

If the bus is being regulated by one or more devices, this field will display the desired regulated voltage (in per unit) the devices are attempting to maintain.

Fault Analysis Load Parameters

This page of the display is only available for buses which have one or more load attached to the bus. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#). It is also possible to define these values as non-zero at a bus where no load exists in the load flow, but it is not usually desirable to do so.

OPF

This tab is only available if you have the Optimal Power Flow (OPF) add-on tool for PowerWorld Simulator. This tab displays the MW marginal cost (Locational Marginal Price) for the bus when performing an OPF solution. The page also breaks down the LMP into its cost components.

Geography

The Geography page of the dialog provides information on the geographic location of the bus on the currently active oneline diagram. Also it provides a conversion tool between [Lat/Lon and UTM Coordinates](#).

Custom

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Bus Voltage Regulating Devices Dialog

The Bus Voltage Regulating Devices Dialog displays summary information about the devices regulating by the selected bus. The regulating devices can be Generators, Switched Shunts and Transformers.

Also the Bus Voltage Regulating Devices Dialog displays a [Case Information Display](#) for the regulating devices. In addition to the information available about the [Generators](#), [Switched Shunts](#) and [Transformers](#) Display, the Bus Voltage Regulating Devices Dialogs Display also shows and allows to modify the Regulating Max and Min per unit voltage, the Device Voltage Target, the Transformer (if any) Regulation Target Type and Device Voltage Target High.

Substation Information (Run Mode)

This dialog is used in the Run Mode to view and modify information associated with a substation record. It displays different information from the [Edit Mode](#) version of the [substation dialog](#). To display it from [Run Mode](#), first select **Aggregation > Substations** in the [Model Explorer](#) to bring up the [Substation Records Display](#). Right-click on the substation of interest and choose **Show Dialog**. The Run Mode Substation Dialog has the following fields:

Substation Number

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

Substation Name and ID

Two alphanumeric identifiers for the substation.

Find By Number

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

Find By Name

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

Find By Sub ID

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

Find...

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

Labels

Clicking this button will open a dialog displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

View All Flows at Substation

Clicking this button will open a [quick power flow display](#) listing the buses contained in the substation.

Information

Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the substation.

Bus Voltages

Summary information on all buses in the substation, including total number of buses, number of dead (disconnected) buses, and minimum and maximum bus voltage and angle within the substation.

Available Gen MW/Mvar Ranges

Total amount of generation increase or decrease available for all generators in the substation.

Buses

The Buses table identifies the buses in the substation, and provides summary information on each.

Gens

The Gens table identifies the generators in the substation, and provides summary information on each.

Loads

The Loads table identifies the loads in the substation, and provides summary information on each.

Switched Shunts

The Switched Shunts table identifies the switched shunts in the substation, and provides summary information on each.

Tie Lines

The Tie Line Table identifies the flows on all of the substation's ties to other substation.

Custom

This page of the dialog can be used to enter notes about the substation. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered for storage with the substation object viewed.

Geography

Displays geographic information about the location of the substation, in Latitude and Longitude. Also it provides a conversion tool between [Lat/Lon and UTM Coordinates](#).

Generator Information (Run Mode)

This dialog is used to view information about each generator in the system. Many fields on this display can also be changed (except in Viewer). Here we describe the Run Mode version of the Generator Information Dialog. The [Edit Mode](#) version is very similar.

Bus Number

Unique number between 1 and 99,999 used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [display filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with [valid display filters](#).

ID

Two character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click the **Find By Number** button.

Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click the **Find By Name** button.

Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

Energized

Indicates if the generator is energized. This is not the same as the Open or Closed status. A generator may be in-service itself (closed) but not be energized, based on the statuses of other devices around the generator. For example, in a full topology model, it is not uncommon for a generator to have a status of Closed, but the generator is not energized if a branch of type Circuit Breaker connecting the generator to the system has a status of Open.

Area Name

Name of the area in which the generator's terminal bus is located.

Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal bus' owner.

Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There additional sections of generator information available from the Run Mode generator dialog:

[Power and Voltage Control](#)

[Generator Cost Information](#)

[OPF](#)

[Fault Parameters](#)

[Owner, Area, Zone, Sub](#)

[Custom](#)

[Stability](#)

Generator Information: Power and Voltage Control

The **Power Control** grouping fields are used to show/change the values associated with the real power output of the generator.

MW Output

Current real power output of the generator.

Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control any time you manually change its output. You could then place the generator back on AGC control by using this dialog.

Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

Loss Sensitivity

Shows how the losses for an area will change for an incremental increase in the generation at the bus. This information is useful in determining the economic dispatch for the generation. The implicit assumption in calculating this field's value is that the incremental change in generation will be absorbed by the system "slack bus." This field cannot be changed.

The **Voltage Control** grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

Available for AVR

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

Use Capability Curve

If checked, the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

Regulated Bus Number

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** from the [Model Explorer](#) to view the [Remotely Regulated Bus Records Dialog](#), which identifies all buses that are being remotely regulated.

Desired Reg. Bus Voltage

Specifies the desired per unit voltage for the generator at the regulated bus. The regulated bus need not be the terminal bus of the generator.

Actual Reg. Bus Voltage

Shows the actual per unit voltage at the regulated bus. If the generator is on AVR and has not reached a reactive power limit, the actual regulated bus voltage should be equal to the desired regulated bus voltage. This field cannot be changed.

Remote Reg %

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100.

Wind Control Mode and Power Factor

The Wind Control Mode effects how generator Mvar limits are treated and can be set to either *None*, *Boundary Power Factor*, *Constant Power Factor* or *Follow Min Mvar Capability Curve*.

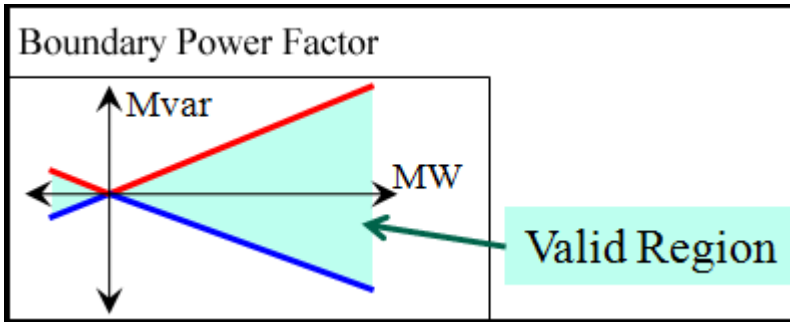
When the mode is *None*, then the generator behaves using the standard settings of the **Min and Max Mvar Output** and **Capability Curve** described above.

For *Follow Min Mvar Capability Curve* mode, the **Mvar output** of the generator is determined by a lookup from the capability curve's Min Mvar value. Essentially the **Max Mvar Output** and the **Min Mvar Output** are then made equal to this value. This provides the ability to make a generator's **Mvar output** any piece-wise linear function of the **MW output**.

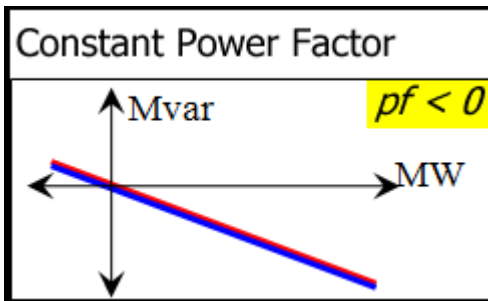
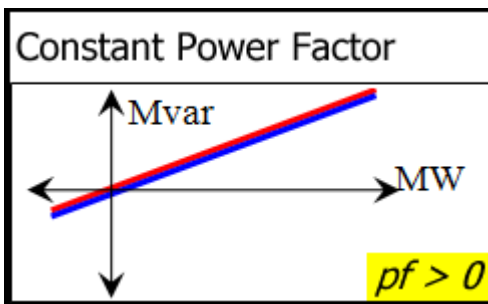
For both the *Boundary* and *Constant* modes, the Mvar limit magnitudes are determined from the actual **MW Output** and the **Wind Control Mode Power Factor** value.

$$\text{Magnitude} = \text{MWOutput} * \tan(\arccos(\text{Power Factor}))$$

For *Boundary* mode, the **Max Mvar Output** is positive and the **Min Mvar Output** is negative. This provides a boundary under which the Mvar must operate.



For *Constant* mode, **Max Mvar Output** and the **Min Mvar Output** are made the same with a positive **Wind Control Mode Power Factor** meaning the limits have the same sign as the actual **MW Output**, and a negative **Wind Control Mode Power Factor** meaning the limits are the opposite sign as the actual **MW Output**, thus the **Mvar output** is a function of **MW Output** and **Power Factor**.



Generator Options: Costs

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

Output Cost Model

Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWhr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form $C(P_{gi}) = (d \cdot P_{gi}^3 + c \cdot P_{gi}^2 + b \cdot P_{gi}) \cdot (\text{fuel cost}) + \text{fixed costs}$ (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

Bid Scale / Shift

Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Marginal Cost (run mode only)

Shows the marginal cost of producing real power at the generator at its current output level, $dC_i(P_{gi})/dP_{gi}$.

ED/OPF Cost (run mode only)

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

Unscaled Cost (run mode only)

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

OPF Reserve Bids

This tab is only available with the [OPF Reserves add-on](#). Detailed information about this tab can be found in the [Generator and Load OPF Reserves Bids](#) topic.

Generator Information: OPF

The fields on this tab display information regarding the generator's participation in an OPF load flow solution.

OPF MW Control

The type of control the generator is allowed during an OPF solution. The generator can be set to No control during OPF, control only if its AGC property is set to Yes, or to always be controlled by the OPF regardless of the AGC status of the generator.

Fast Start Generator

The generator is being treated as a fast start generator during the OPF solution.

Generator MW limits

The MW limits of a generator can be altered in this location if you wish for the generator to use different limits than originally assigned in the load flow case, without actually changing the original values. Simply change the Current Min MW Limit and Current Max MW Limit to alter the limits observed by the generator during and OPF solution.

MW Marginal Cost for Generator's Bus

The OPF solved marginal cost at the generator's terminal bus.

Initial, Final and Delta MW Output

The MW output information for the generator resulting from the OPF run.

Initial, Final and Delta Hourly Cost

The hourly cost information for the generator resulting from the OPF run.

OPF Results

This section shows the results of OPF for the generator.

Generator Options: Owners, Area, Zone, Sub

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

Owners

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

Area Number, Area Name

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

Zone Number, Zone Name

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

Substation Number, Substation Name

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

Load Information (Run Mode)

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. This is very similar to its [Edit Mode](#) counterpart. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with [valid display filters](#). You can use the spin button to cycle through the list of load buses.

Bus Name

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with [valid display filters](#).

ID

Two-character ID used to distinguish multiple loads at a bus. By default, the load id is equal to "1 ." An identifier of '99' is used to indicate an equivalent load.

Find By Number

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

Find By Name

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

Find...

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

Energized

Indicates if the load is energized. This is not the same as the Open or Closed status. A load may be in-service itself (closed) but not be energized, based on the statuses of other devices around the load. For example, in a full topology model, it is not uncommon for a load to have a status of Closed, but the load is not energized if a branch of type Circuit Breaker connecting the load to the system has a status of Open.

Labels

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

Area Number, Area Name

Number and name of the area the load is a member of.

Zone Number, Zone Name

Number and name of the zone the load is a member of.

Owner Number, Owner Name

Number and name of the owner the load is a member of. Loads DO NOT have to be owned by the same owner as the terminal bus.

Substation Number, Substation Name

Number and name of the substation the load is a member of.

Load Information**Base Load Model, Current Load**

The Base Load Model fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields in the Base Load Model section can be changed.

Load Multiplier

The actual load at the bus is equal to the base value multiplied by the corresponding load multiplier. The load multiplier is a value specifying how the load is scaled. The load multiplier depends upon the area load multiplier and the zone load multiplier. See [Load Modeling](#) for more details. The load multiplier value cannot be changed on this dialog.

Bus Voltage Magnitude

Voltage magnitude of the load's terminal bus.

Information about the remaining tabs on the dialog can be found at the following links:

[OPF Load Dispatch](#)

[Custom](#)

[Stability](#)

Branch Information Dialog (Run Mode)

The Branch Information Dialog is used to view information about each transmission line and transformer in the system. You may use this dialog also to change many of the properties of lines and transformers (except in Viewer).

The Run Mode version of this dialog is very similar in content to its [Edit Mode](#) counterpart.

The Branch Information Dialog sports the following fields:

From Bus Number and Name

From Bus number and name. For transformers, the *from bus* is the tapped side.

To Bus Number and Name

To Bus number and name.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

Find By Number

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

Find By Name

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

From and To Bus Area Name

Names of the areas in which the From and To buses are located.

From and To Bus Nominal kV

From and To bus nominal voltage levels.

From and To Bus Voltage (p.u.)

The actual terminal bus voltages of the transmission element, in per unit.

Labels

Clicking on this button will open the [Labels dialog](#) listing all the labels for the selected branch.

Parameters

Status

Current status of the device.

Energized

Indicates if the branch is energized. This is not the same as the Open or Closed status. A branch may be in-service itself (closed) but not be energized, based on the statuses of other devices around the branch. For example, in a full topology model, it is not uncommon for a branch of type Line has a status of Closed, but the branch is not energized if branches of type Circuit Breaker at both ends of the Line are Open.

Per Unit Impedance Parameters

The resistance, reactance, the total charging susceptance (that is, B, not B/2), and the total shunt conductance of the device (in per unit). Magnetizing conductance and susceptance is included if the branch is a transformer.

Line Shunts

Select to view the [Line Shunt](#) Dialog. This dialog is used to create or modify line shunts. Line shunts cannot be created or deleted while in Run Mode. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

MVA Limits

Ratings for the transmission line or transformer in MVA. Eight different limits are allowed.

Flows

These next fields show the actual real and reactive power flow at both ends of the device (because of real and reactive losses these numbers may be different), and its percentage MVA loading. The line losses are summarized as well.

Transformer

The tab is only visible if the branch is a transformer. See the [Transformer Control](#) help for more information on transformer types and controls.

Series Capacitor

This tab is only visible if the selected branch is a series capacitor. If the branch is a series capacitor, the **Is Series Capacitor** box will be checked. In addition, the **Status** field for series capacitors will be enabled, allowing you to change the Bypassed or In Service status of the series capacitor. The series capacitor status IS NOT the same as the branch status of Open or Closed.

OPF

The OPF tab is only visible if you have the OPF ([Optimal Power Flow](#)) add-on tool for PowerWorld Simulator.

Enforce Line Flow Limit

This check box must be checked if the branch limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the branch to violate its branch limits.

Treat Limit as Equality Constraint

If checked, the OPF solution will attempt to solve the load flow while keeping the flow on the branch at its limit.

Maximum MVA Flow

The largest MVA flow value measured on the line, either at the From or To bus.

Present MVA Limit

The limit enforced by the OPF for the branch. This is set in the [OPF constraint options](#), and is related to the original branch limits.

Maximum Percentage

The highest percentage of flow measured on the line, either at the From or To bus.

Limit Marginal Cost

The cost of enforcing the branch MVA limit.

Included in LP

Specifies whether or not the branch flow and limit was included as a constraint in the OPF solution. In general, branches that are not near their limit and do not appear to be changing flow dramatically towards their limit will be ignored in the OPF calculation to speed up the solution. **No** and **Yes** indicate whether or not the OPF process determined that the line needed to be included. The user can initially force the branch to be included or not included with these two fields. By choosing **Always**, the branch will be included in the OPF solution constraints regardless of the propensity of the line to be approaching its limit.

MVA Flow Constraint Status

The constraint status for the line in the OPF solution will be shown here with the corresponding boxes checked by Simulator. These check boxes cannot be changed manually by the user.

From/To Bus MW Marginal Costs

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes an ungrounded wye to ungrounded wye transformer, which has the same model as an open circuit. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

Owner, Area, Zone, Sub, PTDF

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the line's owner is the same than the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the branch dialog.

The area, zone and substation to which the From and To buses belong, are also shown. If a PTDF calculation has been performed, the PTDF values for the viewed line will be displayed as well.

Custom

This page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the branch. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the branch. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Stability

This tab is only visible with the [Transient Stability Add-On](#) tool. Any branch-specific transient stability modeling information is contained on this tab. For more information see the [Transient Stability Data: Object Dialogs](#).

Transformer AVR Dialog

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this dialog, click the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Voltage Regulation (AVR)* option is chosen from the **Automatic Control Type** group.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

Present Regulated Bus Voltage

The present voltage of the regulated bus.

Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the **Regulation Minimum Voltage** and **Regulation Maximum Voltage** fields), the **Voltage Error** field indicates by how much the voltage deviates from the control range.

Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

Present Tap Ratio

The tap ratio of the transformer for the current system state.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on **Power Flow Solution Tab** of the [Simulator Options dialog](#).

Line Drop/Reactive Current Compensation

Typical specification of the bus at which the voltage should be controlled by a transformer is an actual bus in the power system model. However, some control schemes require that the voltage at some fictitious bus a specified impedance away from the transformer be controlled instead. This can be accomplished using the line drop and reactive current compensation control method.

This method requires that the regulated bus for a transformer be specified as one of the transformer's terminal buses. This provides a reference point for specifying the location of the fictitious bus that is the actual controlled point. An impedance must also be specified that will determine the location of the fictitious bus in reference to the selected regulated bus.

Check the **Use Line Drop/Reactive Current Compensation** checkbox to use this control method. Specify the impedance for the location of the fictitious controlled bus using the **Line Drop/Reactive Current Compensation Resistance (R)** and **Line Drop/Reactive Current Compensation Reactance (X)** fields. The impedance should be specified in per-unit.

Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Mvar Control Dialog

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this dialog, click on the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Reactive Power Control* option is chosen from the **Automatic Control Type** group.

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the **Regulation Minimum Mvar Flow** and **Regulation Maximum Mvar Flow** fields, the **Mvar Error** field indicates by how much the flow falls outside the control range.

Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from* bus. The transformer attempts to regulate the reactive flow to fall within this range.

Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

Present Tap Ratio

The transformer's present off-nominal turns ratio.

Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Phase Shifting Dialog

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this dialog, click on the **Change Automatic Control Options** button found with the [Transformer Control options](#), provided that the *Phase Shift Control* option is chosen from the **Automatic Control Type** group.

The **Common Options** tab of this dialog has the following fields:

Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control. Changing the control type will update the dialog to reflect the type of control that is selected.

Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the **MW Error** field indicates by how much the flow violates the regulating range.

Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, the transformer will not change its phase angle. When the regulated MW flow moves outside of this regulation range, Simulator will calculate a new phase angle in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in phase angle. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

Step Size (Degrees)

Phase shift change per step in degrees.

MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the phase angle. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the phase angle changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the **Insert/View Impedance Correction Table** button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click **Insert/View Impedance Correction Table** to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either **OK** or **Save** on the [Branch Information Dialog](#) or [Branch Options dialog](#).

View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that a table must contain at least two points in order to be defined.

The [Time Step Options](#) tab contains control options specific for the Time Step Simulation tool.

Transformer Impedance Correction Tables Dialog

The Transformer Correction Tables Dialog is used to view information about the transformer impedance correction tables. These tables are used on some LTC or phase shifting transformers to model the impedance of the transformer as a function of the off-nominal turns ratio or phase shift. The dialog has the following fields:

Transformer Impedance Correction Table Number

Number of the impedance correction table, between 1 and 63. Use the spin button immediately to the right of this field to step through the list of defined tables. If you have made changes to a particular table, you must click *Save* before moving to another correction table; otherwise, your changes will be lost.

Table Name

The name assigned to the table.

Table Entries

Used to insert/edit/delete the actual entries in the impedance correction table. In the first row, enter either an off-nominal turns ratio for an LTC transformer, or a phase shift in degrees for a phase shifting transformer. The entries in the first row must be entered in strictly ascending form. In the second row, enter the scale factor to apply to the transformer impedance. The transformer's nominal impedance is multiplied by the scale factor to obtain the actual value. Note that at least two columns must be used.

Right-click on the table to invoke its local menu, which allows you to delete and to insert columns. To insert a new column, click on the column before which you want to insert a new column and select **Insert New Point** from the local menu. To delete a column, position the cursor on the column you want to delete and select **Delete Point**.

Table is Used by the Following Transformers

Lists all the transformers in the case that use this impedance correction table. A single table may be used by any number of transformers. To associate a table with a transformer, use the [Transformer AVR Dialog](#) for LTC transformers or the [Transformer Phase Shifting Dialog](#) for phase shifters.

Switched Shunt Information (Run Mode)

The Switched Shunt Information dialog can be displayed by placing the cursor on a switched shunt display object and right-clicking or right-clicking on a switched shunt record in the [Switched Shunt Display](#) and choosing **Show Dialog**. This is very similar to its [Edit mode](#) counterpart. The dialog has the following fields:

Bus Number

Unique number between 1 and 2,147,483,647 (equals 2^{31} minus 1) used to identify the bus to which the switched shunt is attached. This drop down list identifies the buses in the case with switched shunts that also have [valid display filters](#). Use the spin button to step through the list of shunts in the case. Multiple switched shunts are allowed at a bus but only one is allowed to be on automatic control.

Find By Number

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click the **Find By Number** button.

Bus Name

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters. This dropdown box lists the names of all the switched shunt buses in the case with [valid display filters](#).

Find By Name

To find a switched shunt by its name, enter the bus name into the Bus Name field (case insensitive). Then click the **Find By Name** button.

Shunt ID

Since multiple switched shunts are allowed on a single bus, each shunt is identified by a unique Shunt ID.

Find...

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Status

Status of the switched shunt, either Closed (connected to terminal bus) or Open (not connected). On the oneline, the switched shunt can be opened by placing the cursor on the (red) circuit breaker box and clicking, and it can be closed by placing the cursor on the (green) box and again clicking. You can also use this status field to change the switched shunt's status. Note that the switched shunt is only available for automatic control when its status is closed.

Energized

Indicates if the switched shunt is energized. This is not the same as the Open or Closed status. A switched shunt may be in-service itself (closed) but not be energized, based on the statuses of other devices around the switched shunt. For example, in a full topology model, it is not uncommon for a switched shunt to have a status of Closed, but the switched shunt is not energized if a branch of type Circuit Breaker connecting the switched shunt to the system has a status of Open.

Labels

Clicking on this button will open the [Label Manager Dialog](#) listing all the [labels](#) assigned for the selected switched shunt.

Parameters

Nominal Mvar

Amount of reactive power that would be supplied by the switched shunt if its terminal voltage were one per unit (capacitive is positive).

Actual Mvar

Actual reactive power in Mvar being injected into the system by the switched shunt (capacitive is positive). The Actual Mvar field is equal to the Nominal Mvar field multiplied by the square of the terminal bus' per unit voltage.

Nominal MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

Actual MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. The value displayed is the actual real power in MW being injected into the system by the shunt. The Actual MW field is equal to the Nominal MW field multiplied by the square of the terminal bus' per unit voltage.

Control Mode

Information about this option can be found in the [Switched Shunt Control](#) topic.

Control Regulation Settings

Information about these options can be found in the [Switched Shunt Control](#) topic.

Switched Shunt Blocks

Information about these settings can be found in the [Switched Shunt Control](#) topic.

Control Options: Advanced Options

Information about these options can be found in the [Switched Shunt Control](#) topic.

Control Parameters: Time Step Options

The [Time Step Options](#) tab contains control options specific for the [Time Step Simulation](#) tool.

Fault Information

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of steps and blocks.

Owner, Area, Zone, Sub

This tab is used to display or change the generator's owner information, area information, zone, and substation information

Area Number, Area Name

The area number and name to which the switched shunt belongs. The area of the switched shunt can be different than the area of its terminal bus.

Zone Number, Zone Name

The zone number and name to which the switched shunt belongs. The zone of the switched shunt can be different than the zone of its terminal bus.

Owner Number, Owner Name

The owner number and name to which the switched shunt belongs. A switched shunt will belong to the same owner to which its terminal bus belongs.

Substation Number, Substation Name

The substation number and name to which the switched shunt belongs. A switched shunt will belong to the same substation to which its terminal bus belongs.

Custom

This page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the switched shunt. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Stability

This tab is only visible with the [Transient Stability Add-On](#) tool. Any switched shunt-specific transient stability modeling information is contained on this tab. For more information see the [Transient Stability Data: Object Dialogs](#).

Zone Information (Run Mode)

This dialog is used in the Run Mode to view and modify information associated with a zone record. It displays different information from the [Edit Mode version of the zone dialog](#). To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Right-click on the zone of interest and choose **Show Dialog**. The Run Mode Zone Dialog has the following fields:

Zone Number, Zone Name

Number and name of the associated zone. Use either the combo box or the spin arrows to view the different zones.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Labels

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels or aliases assigned for the selected zone.

The rest of the Run Mode Zone Dialog is divided into multiple pages:

Information

Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the zone.

Generation AGC Range

These two fields show the total available MW reserve for generators in the zone that are on AGC and have nonzero participation factors. In other words, these fields show the total MW by which the generation in the zone can be increased or decreased using only generation that is presently on-line. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

Tie Lines

Zone Tie Lines

The Zone Tie Line Table identifies the flows on all of the zone's ties to other zones.

To determine the total loss for a zone, losses for tie-lines between zones are assigned to the zone in which the terminal bus that is NOT the metered bus is contained. The **Metered End** field for a branch determines which of the terminals is metered.

OPF

Average LMP for Zone

The computed average locational marginal price of all buses contained in the zone.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the zone.

Minimum LMP

The minimum locational marginal price of all the buses in the zone.

Maximum LMP

The maximum locational marginal price of all the buses in the zone.

Reserve Requirement Curves

These options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Zone Buses, Zone Gens, Zone Loads, Zone Switched Shunts

These pages show the case information display with the buses, generators, loads and switched shunts assigned to the zone.

Custom

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.

Using Labels for Identification

Most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) may have an alternative names assigned to them. These alternative names are called labels. Labels allow you to refer to equipment in the model in a way that may be unique to your organization. Labels may thus help clarify which elements are described by a particular set of data, especially when the short names employed by the power system model prove cryptic.

Furthermore, since labels are likely to change less frequently than bus numbers, and since a label must, by definition, identify only one power system component, they may function as an immutable key for importing data from [auxiliary files](#) into different cases, even when bus numbering schemes change between the cases. Labels must be unique for devices of the same type, but the same label can be used for a device of a different type.

Information dialogs corresponding to buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces feature a button called **Labels**. If you press this button, the device's Label Manager Dialog will appear. The [Label Manager Dialog](#) lists the labels associated with the device and allows adding, deleting, and modifying labels. The **Labels (All)** field (variablename = LabelsAll) in a [Case Information Display](#) lists all of the labels that have been associated with a device. The device's primary label is the one that is listed first in this field. This field lists all labels assigned to a device as a comma-delimited string. Even though multiple labels may be assigned to a device, any label can be used to import data from [auxiliary data files](#). The Label Manager Dialog can be used to assign labels or the string of labels can be entered directly in the **Labels (All)** field.

Labels can be used to map data from an auxiliary data file to a power system device. Recall that [auxiliary data files](#) require you to include a device's key fields in each data record so that data may be mapped to the device. Labels provide an alternative key. Instead of supplying the bus number to identify a bus, for example, you can supply one of the bus's labels. The label will enable Simulator to associate the data with the device associated with that label. This mechanism performs most efficiently when the primary label is used, but other labels will also provide the mapping mechanism. The **Label (for use in input from AUX or Paste)** field (variablename = Label) is used for importing data using labels and is blank when viewing in a case information display. Keep in mind that all devices read via an auxiliary file using the label field should have a non-blank label. Otherwise, information for that device will not be read. Even if the primary or secondary key fields are provided with the device, as long as the label field is present, that is the only field that will be used to identify the device. New devices cannot be created by simply identifying them by label. Either the primary or secondary key fields must be present to create a new device and the label field should not be present.

Again, it is important to remember this: a single power system device may have multiple labels, but each label may be associated with only one device of a particular type. This is the key to enabling data to be imported from an auxiliary file using labels.

Saving Auxiliary Files Using Labels

All devices that can be identified by labels will have the **Labels (All)** and **Label (for use in input from AUX or Paste)** fields available in their case information displays. In order to save auxiliary files that identify devices by label, the two label fields should be added to the case information display prior to saving the data in an auxiliary file. Because the **Label (for use in input from AUX or Paste)** field will be blank when saved in the auxiliary file, this field must be populated with one of the labels in the **Labels (All)** field before loading the auxiliary file back in. Keep in mind that devices with blank labels cannot be identified when loading in an auxiliary file, so avoid saving auxiliary files by label if all devices do not have labels.

Many devices require [SUBDATA](#) sections. These sections have custom formats specific to the type of information that they contain. When saving auxiliary files with devices that require SUBDATA sections, the user can choose to use primary or secondary key fields or labels to identify devices in the SUBDATA sections. The user will either be prompted when saving the devices, or there is an option to change the key field to use when saving subdata sections on the PowerWorld Simulator Options dialog under the [Case Information Displays](#) category. When choosing to use labels, if a device has a label, it will be used. If it is a device that can be identified by buses and bus labels exist, bus labels will be used. Finally, if the device does not have a label and the buses do not have labels, the primary key for the device will be used for identification.

Devices that have SUBDATA sections that contain other devices that can be identified by labels include: contingencies, interfaces, injection groups, post power flow solution actions, and owners.

The setting to choose which identifier to use for the SUBDATA sections does not just apply to SUBDATA sections. Often when saving groups of options, this setting will apply to everything being saved with those options and not just the SUBDATA sections. This includes contingency options, ATC options, limit monitoring settings, and PVQV options. In these cases, there will be a prompt asking the user to decide which identifier to use in the auxiliary file.

Loading Auxiliary Files SUBDATA Sections Using Labels

The various SUBDATA sections that represent references to other objects can also be read using labels. Examples include contingencies, interfaces, injection groups, post power flow solution actions, and owners. When reading a SUBDATA section such as this, PowerWorld makes no assumption ahead of time about what identification was used to write this SUBDATA section. Instead, an order of precedence for the identification is as follows

	Identification	Explanation	Example
1	Key Fields	assumes that the strings represent Key Fields	BRANCH 8 9 1
2	Secondary Key Fields	assumes that the strings represent Secondary Key Fields	BRANCH Eight_138 Nine_230 1
3	Labels for component objects	the key/secondary key fields for some objects consist of references to other objects. An example of this is the BRANCH object that is described by the From Bus, To Bus, and Circuit ID. This assumes that labels of the component objects are used.	BRANCH Label8 Label9 1
4	Labels	Assumes that the string represents one of the Labels of the object	BRANCH LabelForBranch

Special Use of Labels

There are a few special cases where objects have fields that identify other devices. These devices can be identified by label but not in the conventional means because the label field applies to the object that contains the device and a SUBDATA section is not necessary. These special cases include: (Note all fields given below are by variable name because the use of labels is most relevant with auxiliary files.)

ATC Scenarios

[ATC Scenario](#) change records usually contain primary key fields to identify the devices that should be adjusted during the scenario. If using labels, these primary key fields will be replaced with a single **Label** field. The use of this field is different because the Label field refers to the device in the change record and not to the change record itself. When labels are used with ATC scenarios, device labels only can be used. Bus labels cannot be used to identify devices for which no label exists but a bus label does.

ATC Scenarios are saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

ATC Extra Monitors

[ATC Extra Monitors](#) identify either branches or interfaces to monitor during the ATC analysis. These devices are identified in the **WhoAmI** field of ATC Extra Monitor records. Usually, the **WhoAmI** field is a special format that contains key field tags. Optionally, this field can use the label of the device for the extra monitor. If the device label is not available, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

ATC Extra Monitors area saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

Model Conditions

Devices in [Model Conditions](#) are usually identified by the **WhoAmI** field which is in a special format that contains key field tags. Optionally, this field can use the label of the device. If the device label does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Model Expressions

[Model Expressions](#) contain Model Fields. Model Fields are identified by the **WhoAmI** fields in the Model Expressions. Usually, the WhoAmI fields are in a special format that contains key field tags. Optionally, these fields can use the label of the device associated with the Model Field. If the device does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Bus Load Throw Over Records

[Bus Load Throw Over Records](#) are used with contingency analysis. These records have an option to identify the bus to which the load will be transferred by either number or name_kV combination. If choosing to identify objects by label, the **BusName_NomVolt:1** field will contain the label of the bus instead of the name_kV combination.

Bus Load Throw Over Records will be saved in an auxiliary file if choosing to [Save](#) settings on the Contingency Analysis dialog.

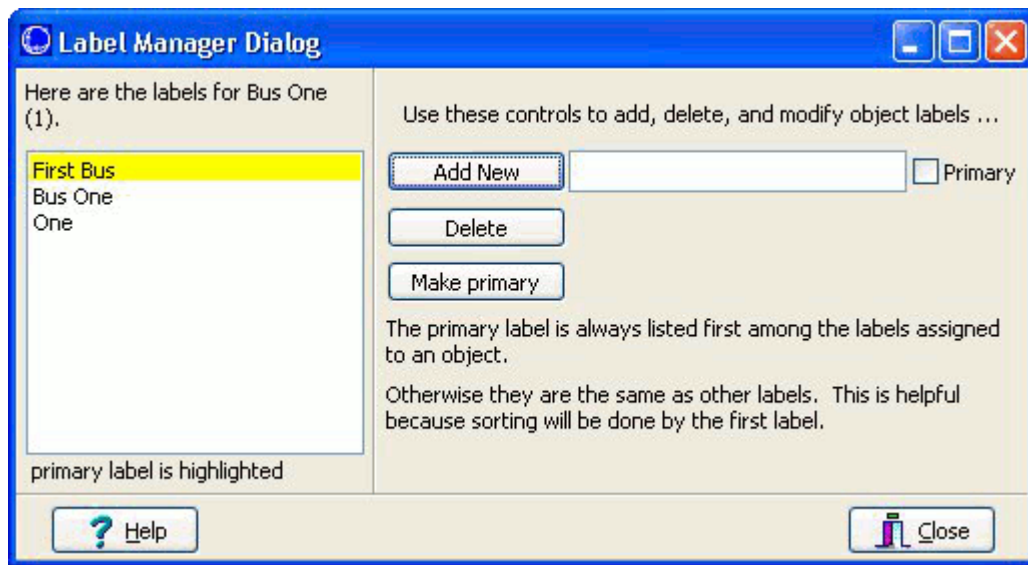
Injection Group Participation Points

All participation points and the injection groups to which they belong can be listed on the [Injection Group Display](#). Load, generator, and shunt devices that can be assigned to a participation point must be identified by bus and ID. The bus can be identified by either the number or name. When identifying by name, the **BusName_NomVolt** field is used to provide the name_kV combination for the bus. If choosing to identify devices by label, this field instead will contain the label of the device. If the device does not have a label but the bus does, the bus label will be used instead in conjunction with the ID of the device. Even if the device does contain a label, the ID field must be included in any auxiliary file that is going to be loaded because it is a key field. Injection groups can be included in other injection groups. Injection groups can be identified by label, even though this is not a normal thing to do. If any injection groups have labels and these injection groups are included in other injection groups, their labels will also appear in the BusName_NomVolt field. If they do not have labels, they will be identified by the injection group name that appears in the PPntID field.

Label Manager Dialog

The Label Manager Dialog allows the creation and modification of [labels](#) to identify power system devices. Labels can be used for most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) as alternate names for these devices. See the [Labels](#) topic for more information on how labels can be used.

The Label Manager Dialog can be accessed by clicking the Labels button on any of the information displays for objects that allow labels.



List of Labels

Multiple labels can be assigned to a single object. The list on the left of the dialog shows all of the labels that have been assigned with the primary label listed first and highlighted in yellow.

Add New

Click this button to add a new label after entering the new label in the box to the right of the button. If this new label should be the primary label, check the **Primary** box. Labels must be unique for each type of object. That means that there cannot be two buses labeled *One*. However, there could be objects of different types with the same label. There could be a bus and a generator both labeled *One*. If attempting to label an object with a label that has already been used by the same object type, the label will be assigned to the current object and removed from the object to which it was previously assigned.

Delete

Click this button to delete a label after first selecting the label in the list of labels.

Make Primary

Click this button to make a label the primary label after first selecting the label in the list of labels. The primary label will be moved to the top of the list. The primary label will always appear at the top of the list and will be used for sorting when sorting by label.

Latitude/Longitude and UTM Coordinates Conversion

This tool is used to convert between Latitude/Longitude and UTM Coordinates in a particular Bus. The Tool is located in the Geography Tab of a Bus or Substation Dialog in the Run or Edit mode.

Device Info | **OPF** | **Geography** | Custom

Geographic Location

Latitude (degrees) UTM North/South

Longitude (degrees) UTM Longitude Zone

Datum UTM Northing

UTM Easting

Note: Any latitude and longitude pair where $\text{abs}(\text{latitude}) > 90$ or $\text{abs}(\text{longitude}) > 180$ will be considered invalid.

Latitude (degrees)

Entry for the latitude information that is stored with the chosen bus. The Latitude entry can also be entered manually.

Longitude (degrees)

Entry for the longitude information that is stored with the chosen bus. The Longitude entry can also be entered manually.

Datum

Entry for the Datum (model of the shape of the earth) information that is stored with the chosen bus. The Datum entry can also be entered manually. Example: WGS-84.

Convert Lat/Lon to UTM Coord.

Press *Convert Lat/Lon to UTM Coord.* to convert from Lat/Lon coordinates to UTM coordinates.

UTM North/South

Entry for the side of the hemisphere: North or South. The UTM North/South entry can also be entered manually.

UTM Longitude Zone

Entry for the zone information. Also the UTM Longitude Zone entry can also be entered manually.

UTM Northing

Entry for the Northing information. Also the UTM Northing entry can also be entered manually.

UTM Easting

Entry for the Easting information. Also the UTM Easting entry can also be entered manually.

Convert UTM to Lat/Lon Coord.

Press *Convert UTM to Lat/Lon Coord.* to convert from UTM coordinates to Lat/Lon coordinates.

For more information on the Map Projections or conversion between UTM and Latitude/Longitude coordinates please go to <http://www.uwgb.edu/dutchs/usefuldata/utmformulas.htm> or:

Snyder, John P., Map Projections - A Working Manual, U. Geological Survey Professional Paper 1395, United States Government Printing Office, Washington, D.C.: 1987.

Area Information

The Area Information Dialog shows information about each area in the system. It displays an area's load, generation and losses; the area's scheduled interchange with other areas; options for controlling the area's generators, transformers, and shunts; the flows on its tie lines; and its operating cost information. You may view this dialog by doing any of the following:

- Select Areas from the [Model Explorer](#). This displays the [Area Display](#). Select the desired area and click the Show Dialog option on the [case information toolbar](#), or right-click on the record corresponding to the desired area to bring up the display's local menu and choose Show Dialog.
- Right-click in an empty portion of the oneline near a bus in the area of interest to display the [oneline's local menu](#). Select Area Information Dialog.
- While in Run Mode, right-click on an [Area Display Object](#) that represents an area and choose Information Dialog. Note: Doing this while in Edit Mode will show the [Area Display Options](#) dialog instead of the Area Information Dialog.

The Area Information Dialog contains the following information:

Number

A drop-down box that specifies the area number. Select an area from the drop-down box, or use the spin button to cycle through the list.

Find By Number

To find an area by its number, type the number in the **Number** field and click this button.

Name

A drop-down box that specifies the area's alphabetic identifier, which may be any length in Simulator. When saving area names to other load flow formats, the names are truncated at the maximum character length supported by that format.

Find By Name

To find an area by its name, type the name in the **Name** field and click this button.

Find...

If you do not know the exact area number or name you are looking for, you can click this button to open the [advanced search engine](#).

Super Area

To associate an area with a particular super area, either select an existing super area from the drop-down box, or type the name of a new super area.

Area Control Options

Select to change the area's method of [Automatic Generation Control \(AGC.\)](#)

Labels

To assign alternative identifying labels to the area, click the Labels button.

The rest of the Area Information Dialog is divided into nine pages of controls:

- [Info/Interchange](#)
- [Options](#)
- [Area MW Control Options](#)
- [OPF](#)

- [Tie Lines](#)
- [Buses](#)
- [Gens](#)
- [Loads](#)
- [Custom](#)

Area Information: Buses

The Area Buses grid shows a list of all buses that are in the area. The grid displays the bus number, bus name and other bus information. The Area Buses display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Gens

The Area Gens grid shows a list of all generators that are in the area. The grid displays the bus number, bus name, generator ID and other generator information. The Area Gens display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Loads

The Area Loads grid shows a list of all loads that are in the area. The grid displays the bus number, bus name, load ID and other load information. The Area Loads display is a [Case Information Display](#) and has options available to these types of displays.

Area Information: Area MW Control Options

Set Participation Factors...

Press this button to open the [Generator Participation Factors Dialog](#), which gives you control over how the participation factor for each generating unit is defined. If you decide not to prescribe participation factors using this button, the existing participation factors defined with each generator will be used. This button will be enabled only if the area has been set to Participation Factor area control.

Area Slack Bus Number

Identifies which bus to model as the area's slack bus.

Injection Group Area Slack

An additional area generation control option that can be utilized is the ability for Simulator to use a defined [injection group](#) as the sink for Area Slack control. When an injection group is selected, and the area is set on Area Slack control, any needed injection changes in the area will be accounted for by the devices in the injection group.

Some typical injection group options are also available here for handling the injection group dispatching during the Injection Group Area Slack Control. These options include:

Allow only AGC gen/load to vary:

Only generators and loads in the injection group that are designated as available for AGC control will be allowed to participate in injection changes.

Enforce generator MW limits (ignore case and area options):

Generators in the injection group can be designated to enforce their MW limits independent of the global case and area options for generator MW limit enforcement.

Do not allow negative loads:

If loads are decreasing demand as part of an injection group area dispatch, you can specify whether or not those loads should be allowed to go below 0 demand.

How should reactive power load change as real power load is ramped?

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

Area Information: Info/Interchange

The Info/Interchange Tab serves as an accounting sheet for flows into and out of an area. It houses the following controls:

Load, Generation, Shunts, Losses, Interchange

These read-only fields express the total real and reactive load, generation, shunt compensation, losses, and interchange for the area.

Interchange

Interchange in MW between the area and all other areas (exporting power is positive). To prescribe an interchange with a specific area, simply enter the amount of power in MW to export to that area in the MW Export column. To specify an import, enter a negative value. If the recipient of the power is unknown, you may enter the net total in the **Unspecified MW Interchange** field found above the list of transactions. The net value of the MW imports and exports is displayed in **Transaction MW**.

Check the box **Only Show Areas with Nonzero Interchange** to display only those areas whose interchange is different from zero.

By default, one interchange record is initially defined between each area. The total interchange between two areas can be managed using this single record. However, it is possible to define multiple interchange records between the same areas. To do so, you can right-click in the interchange table and select Insert from the local menu. This will open the [Transaction Dialog](#), which will allow you to insert a new transaction. Transactions between the same two areas must have unique transaction IDs. Once you have multiple transactions defined between the same two areas, you can choose which transactions are enabled using the Enabled property in the Base Interchange table. This field is a toggleable field, meaning you can double-click in the field to toggle its value.

The total interchange defined for an area will be displayed in the summary field labeled **Interchange** in the area summary totals on the left side of the page.

AGC Tolerance

The MW tolerance is used in enforcing area interchange. When the absolute value of the ACE is less than this value, Simulator considers the area interchange constraint to be satisfied.

ACE (Area Control Error)

Current area control error (ACE) for the area in MW. Note that, when the constant frequency model is used, $ACE = \text{area generation} - \text{area load} - \text{area losses} - \text{scheduled area interchange} - \text{area shunt MW}$.

Area Has Multiple Islands

This is an informational field that cannot be changed. If checked, then Simulator has detected that the area has devices in separate [electrical islands](#) in the system. This is important because it is sometimes impossible for an area using some form of generation control ([AGC](#)) to solve for the generation dispatch in the entire area ($\text{Area Generation} = \text{Area Load} + \text{Area Losses} + \text{Area Interchange} + \text{Area Shunt MW}$) when the area is not entirely within the same electrical island. If Simulator is unable to solve for areas that span multiple islands, the AGC status of the area will automatically be set to "Off AGC" and a warning message will be written to the Simulator [message log](#).

Area Information: Options

Report Limit Violations

If checked, limit violations for this area are reported. Limits violations are reported in the [Limit Violations List](#).

Generation AGC Values

These fields indicate the total available AGC range for all the generators in the area **that are specified as being on AGC and have nonzero participation factors**. That is, these fields show the total amount the generation in the area that can be increased or decreased using the presently on-line generation. This may be referred to as the *spinning reserve*. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

Load MW Multiplier Value

This value is used for scaling the area MW load. The base load remains unchanged.

Load MVAR Multiplier

This value is used for scaling the area MVAR load. The base load remains unchanged.

Automatic Control Options

The Automatic Control Options section provides a convenient mechanism to enable or disable automatic control of switched shunts and transformers in the area.

This section also provides a check box for setting if the generators in the area should enforce their Generator MW Limits or not. Typically this should be checked. Note that even if unchecked, the [OPF](#) routine end Economic Dispatch will ignore this setting and will always enforce generator MW limits during an OPF solution. Settings for enforcing MW limits at individual generators and globally for the case will also be ignored if using OPF or Economic Dispatch control.

Include Loss Penalty Factors in ED

If this box is unchecked, then the economic dispatch for the area is calculated assuming that the area is lossless. Otherwise, the solution will incorporate losses when computing the economic dispatch. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

Economic Dispatch Lambda

The lambda value calculated during the economic dispatch computation. This field is valid only when the area is on economic dispatch control.

Area Information: Tie Lines

The Tie Lines display shows the flow on all of the tie lines for the selected area. This display is a [case information display](#) and contains many of the options available to these types of displays.

The default fields that are shown in the grid identify each line and show the flow on each line.

Near Area Name, Near Area Number

These fields identify the area name and number of the tie line terminal located in the selected area.

Near Number, Near Name

These fields identify the bus number and name of the tie line terminal located in the selected area.

Far Area Name, Far Area Number

These fields identify the area name and number of the tie line terminal located in the other area.

Far Number, Far Name

These fields identify the bus number and name of the tie line terminal located in the other area.

Ckt

This is the circuit identifier for the tie line.

Meter MW, Meter Mvar

These are the real and reactive power **flowing on the tie line from the selected area to the other area**. (Real and reactive power flowing on line from the Near bus to the Far bus.)

Status

This field gives the status of the tie line as *Open* or *Closed*.

MW Loss, Mvar Loss

These fields show the real and reactive power losses on the tie line.

To determine the total loss for an area, losses for tie-lines between areas are assigned to the area in which the terminal bus that is NOT the metered bus is contained. The **Metered End** field for a branch determines which of the terminals is metered.

Area Information: OPF

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for an area. It houses the following information:

Average LMP for Area

The computed average locational marginal price of all buses contained in the area.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the area.

Min/Max LMP

The minimum and maximum locational marginal price of all the buses in the area.

Total Generator Production Cost (Scaled)

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

Total Generator Unscaled Production Cost

The total unscaled generator production cost, based on the original generator cost or bid curves.

Total Generator LMP Profit

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

Cost of Energy, Loss, and Congestion Reference

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.

Reserve Requirement Curves

These options are only available with the [OPF Reserves add-on](#). More information about these options can be found in the [Area and Zone OPF Reserve Requirement Curves topic](#).

Area Information: Custom

The Custom page of the [Area Information](#) dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for areas. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the Area Information dialog is simply a location to log information about the area. To log information about the area, simply switch to the Custom page on the dialog, and start typing your information or comments about the area in the memo box.

Area Field Options

Area field objects are used to show different values associated with areas and the system. This dialog is used to view and modify the parameters associated with these fields.

Area Number

Area number associated with the field. When you insert fields graphically, this field is automatically set to the area number associated with the closest bus on the oneline. With most types of area fields, an *Area Number* of 0 is valid and defines the field as showing values for the entire system.

Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Other Area Number

Some of the fields, such as **MW Flow to Other Area**, require that a second area be specified. If applicable, enter the second (other) area here.

Other Area Transaction ID

Since it is now possible to have more than one base transaction defined between the same two areas, base transactions must now also have a unique ID to distinguish between transactions among the same two areas. If you are displaying a field that pertains to the display of the scheduled flow between areas, the ID of the transaction in question will also need to be entered here.

Delta MW per Mouse Click

This value is used only with the **Sched Flow to Other Area** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Area**, a spin button is shown to the right of the area field. When the up spin button is clicked, the flow to the other area is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Field Value

Shows the current output for the area field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Area** field type only, you can specify a new value in MW. Exports are assumed to be positive.

Anchored

If checked, the area field will be anchored to a corresponding [Area Object](#). If the area object is moved on the diagram, the text field will move with it.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of bus field to show. The following choices are available:

Name Area name (eight characters maximum). The *Area Number* field must correspond to a valid area.

Number Area number (1 - 999). The *Area Number* field must correspond to a valid area.

MW Load, Mvar Load If the area number is nonzero, then these fields show Total MW or Mvar load for the area. If the area number is zero, these fields show the total load in the entire system.

MW Generation, Mvar Generation If the *Area Number* is nonzero, then these fields show Total MW or Mvar generation for the area. If the *Area Number* is zero, these fields show the total generation in the entire system.

MW Losses, Mvar Losses If the *Area Number* is nonzero, then these fields show Total MW or Mvar losses for the area. If the *Area Number* is zero, these fields show the total losses in the entire system.

ACE (MW) Area Control Error in MW for the area. The *Area Number* field must correspond to a valid area.

Hourly Cost (\$/hr) If the *Area Number* is nonzero, then this field shows the hourly cost for the area. If the *Area Number* is zero, these fields show the hourly cost for the entire system.

Total Cost (\$) If the *Area Number* is nonzero, this field shows the total cost incurred by the area since the beginning of the simulation. If the *Area Number* is zero, this field shows the total cost incurred throughout the system since the beginning of the simulation.

MW Flow to Other Area,

Mvar Flow to Other Area Total MW or Mvar flow from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total MW or Mvar exports.

Sched. Flow to Other Area Scheduled MW transaction from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field, and with the Transaction ID given in the *Other Area Transaction ID* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total scheduled MW transactions. If the *Delta per Mouse Click* value is nonzero, you can use the spin arrows on the oneline to change this value. Also, you may directly enter a new value in the *Field Value* field.

Load Schedule Multiplier (MW Only) Indicates the current value of the MW multiplier applied to the area's loads.

AGC Status Displays the AGC status of the area.

Select a Field Choose from all possible information fields that can be displayed for areas.

Select **OK** to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

Super Area Information Dialog

The Super Area Information Dialog displays information pertaining to [Super Area Display](#). It summarizes the super area's real and reactive load and generation, scheduled transactions, and constituent areas. It also allows you to manage the list of areas to include in the super area and to designate how its generation should be controlled. To display the Super Area Information Dialog, right-click on the [Super Area Records Display](#) and select **Show Dialog** from the resulting local menu.

The Super Area Information Dialog has the following fields:

Area in Super Area

Name

The list of all super areas that have been defined in the case are listed in this dropdown box. Select one of the super areas to display its information, or type a new name in the box and click **Add New** to define a new super area.

Rename

Click this button to change the name of the super area specified in the Name field.

Save

Saves the super area of the name specified in the list of super areas.

Delete

Click this button to delete the super area specified in the Name field.

Area in Super Area

Areas in Super Area

Lists the areas contained in the super area. Right-click on a row and select **Remove** to remove the area from the super area.

Super Area Control Options

Specify the type of generation control to employ for the super area. The super area may be removed from area control or employ Participation Factor or Economic Dispatch control. See [Area Control](#) for details on these types of generation control.

An additional type of control available to Super Areas is to **Use Area Participation Factors**. If the super area is set to participation factor control, and this option is NOT checked, then the generators within the super area respond by redispatching to meet the entire super area generation change based on their own participation factors.

However, if this option IS checked, then an additional level of participation control complexity is added for the super area. First, the total super area generation change is divided across each area forming the super area, according to the participation factors of the AREAS, as set in the grid on this dialog. Once the total generation change has been determined for each area within the super area, then the generators within each individual area are dispatched using [area participation factor control](#) to meet each specific area's determined generation change.

Use Area Participation Factors

If the super area control is set to Participation Factor control, you can specify participation factors for each area in the list of areas that indicate how each area should contribute towards changes in generation for the super area. For example, if you have three areas in the super area and you set the participation factors to 1, 1 and 2, the three areas will contribute to the total change in

generation in the super area by ratios of 25%, 25%, and 50%, respectively. The contribution of each area is then further divided among the generators within each area by the participation factors of the individual generators.

This option will be applied when calculating [TLR](#), [PTDF](#), [voltage](#), and [ATC](#) sensitivities. When calculating these sensitivities, the assumption is that the super area is on participation factor control even if the control method is something other than this. If this option is selected, it will be enforced regardless of the control method when calculating these sensitivities.

New Area Name

Use the dropdown box to select an area to add to the super area. Click the **Add New Area by Name** to add the selected area to the super area.

New Area #'s

Enter in a list of area numbers separated with dashes or commas and click **Add New Areas by Number** button to add the areas to the Super Area

Summary Information

ACE Total (MW)

The total area control error for the super area. It is calculated by summing the ACE for all areas comprising the super area.

ACE Tolerance (MW)

The Area Control Error tolerance observed for the super area during a load flow solution.

Total Scheduled Transactions

Lists the total scheduled import or export for the super area. It is computed by summing the scheduled interchange for all areas comprising the super area. Exports are positive.

Lambda

The marginal cost associated with the super area. Lambda is valid only for super areas that are on economic dispatch control.

Hourly Cost

Current average hourly cost for the super area in \$/hour.

Load, Generation, Shunts, Losses, Interchange

The Load and Generation section of the Super Area Information Dialog accounts for the real and reactive power flows into, out of, and within the super area. Each quantity is computed by summing over the areas comprising the super area.

OPF

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for a super area. It houses the following information:

Average LMP for Area

The computed average locational marginal price of all buses contained in the area.

LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the area.

Min/Max LMP

The minimum and maximum locational marginal price of all the buses in the area.

Total Generator Production Cost (Scaled)

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

Total Generator Unscaled Production Cost

The total unscaled generator production cost, based on the original generator cost or bid curves.

Total Generator LMP Profit

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

Cost of Energy, Loss, and Congestion Reference

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.

Custom

This page of the dialog can be used to enter notes about the super area. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered on this page in the form of floating point, integer, and string fields. These custom fields are stored with the super area and can be used for various tasks in Simulator.

Super Area Field Information

Super area field objects are used to show different values associated with super areas. This dialog is used to view and modify the parameters associated with these fields. Note that the super area name itself cannot be changed on this dialog.

Super Area Name

Select the name of the super area for which you are inserting or viewing information of a super area field.

Find...

If you do not know the exact super area name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

Anchored

Toggles if the field should be anchored to an associated super area object on a diagram.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of super area field to show. The following choices are available:

Name Super area name (eight characters maximum). The super area name field must correspond to a valid super area.

MW Load, Mvar Load Total MW or Mvar load for the super area.

MW Generation, Mvar Generation Total MW or Mvar generation for the super area.

MW Losses, Mvar Losses Total MW or Mvar losses for the super area.

ACE (MW) Area Control Error in MW for the super area. The super area name field must correspond to a valid super area.

MW Exports Total MW exports for the super area.

Hourly Cost (\$/hr) The hourly cost for the super area.

AGC Status Displays the AGC status of the super area.

MW Marginal Cost (\$ / MWhr) MW marginal cost for the super area.

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

Interface Information

The Interface Dialog is used to create, modify, or delete interface records in both the Edit and Run Modes. This dialog has the following controls:

Interface Name

An alphanumeric identifier for the interface of up to 24 characters. Use the dropdown box or the spin button to navigate through the list of existing interface records.

Interface Number

A numeric identifier for the interface, between 1 and 32,767.

Find Interface...

If you do not know the exact interface name or number you are looking for, you can click this button to open the [advanced search engine](#).

Add New Interface

Click the Add New Interface button to define a new interface from the Interface Dialog. When you click this button, the **Interface Name** field and **Interface Elements** table are cleared, requiring you to enter a new name and new elements.

Delete Interface

Select this button to delete the currently displayed interface. Once the interface record has been deleted, the **Interface Name** field displays the previous interface record, if any. If there are not previously defined interface records, Simulator will close the dialog.

Interface Limits

Specify the possible limits for net interface flow. As for transmission lines and transformers, up to eight distinct limits can be specified for interfaces. Which limit set is used can be controlled from the *Limits Tab* of the [Simulator Options Dialog](#) or the [Line and Transformer Limit Violations Display](#).

Monitoring Direction

This area allows the user to change the flow monitoring direction. Clicking on the interface in the **Interface Elements** area will bring up the **Interface Elements Dialog**. This is where the default direction is set. To keep this default direction, choose FROM → TO, to change this direction, choose TO → FROM, and to monitor both directions, choose Both Directions.

Noncontingent MW Flow Contribution

Indicates the present net flow through the interface elements, not including affects of contingencies, in MW.

Contingent MW Flow Contribution

The amount of flow that would be added to the interface flow if the contingency defined with the interface were to occur, in MW.

Total MW Flow

The sum of the non-contingent MW flow and Contingent MW flow, for the total MW flow considered on the interface, in MW.

PTDF Value (%)

The [Power Transfer Distribution Factor](#) for the interface, if calculated.

Interface Elements

The Interface Elements Table lists each element comprising the interface. If Simulator is in Run Mode, the table will also show the present flow through each element. To edit or delete an existing element in the table, click on it to bring up the [Interface Element Dialog](#). Use the Interface Element Dialog to modify or delete the element.

Insert New Element

Click the Insert New Element Dialog to add a new element to the interface using the [Interface Element Dialog](#).

Click **OK** to save any changes you have made and to close the Interface Information Dialog. Click **Save** to save your changes but to leave the dialog open so that you can view and modify other interface records. Click **Cancel** to close the dialog without saving your latest change.

Element Identifiers

Choose whether to show the interface descriptions using the interface names, the interface numbers, or a combination of both.

OPF

The OPF page will only be visible if you have the Optimal Power Flow add-on for PowerWorld Simulator. This page contains interface information relating to performing an OPF solution.

Enforce Interface Flow Limit

If checked, the Interface limit will be checked for enforcement during an OPF solution.

Treat Limit as Equality Constraint

If checked, the OPF routine will try and maintain the flow on the interface at its limit value. Otherwise the limit will be treated as the maximum for an inequality constraint.

MW Flow on Interface

The actual MW flow on the interface.

Present MW Limit

The currently defined limit for the interface.

Percentage of Limit

The MW flow on the interface as a percentage of the limit.

Limit Marginal Cost

The incremental cost of maintaining the flow on the Interface at its limit value.

MW Flow Constraint

The constraint status of the interface in the OPF solution is shown here with the corresponding boxes checked by Simulator. These boxes can not be changed by the user.

Interface Element Information

The Interface Element Dialog is used to redefine or to add the individual elements comprising an [interface](#). Individual lines or transformers, inter-area ties, inter-zone ties, line contingencies, DC lines, injection groups, generators, and loads may make up an interface. The Interface Element Dialog allows you to add all three varieties of interface elements to an interface.

The Interface Element Dialog comes in two very similar forms, depending upon how it was invoked. The dialog may be called from the [Interface Information Dialog](#) by clicking on either the Interface Elements Table or the Insert New Element button.

The Interface Element Dialog contains the following controls:

Element Type

Specifies the type of interface element being investigated or added. Interface elements can be any line or transformer or, more specifically, they can be groups of lines and transformers that tie two areas or two zones together. Interfaces can also contain line contingencies, DC lines, generators, loads, and injection groups.

When the selection for Element Type changes, the available **Element Identifiers** change to allow you to pick the appropriate elements of that type.

Element Identifiers

Depending upon the **Element Type** selection, different element identifiers are required to designate the element to add to the interface. You can search through the list of identifiers to find the particular elements you wish to include. Note that the flow direction on transmission line, transformer, and DC line elements will be dependent on which end of the line you choose as the near bus. The flow will always be measured in the direction of near bus to far bus. The flow will be positive when flowing from near to far, and negative when flow from far to near. In addition, line elements also have an additional setting labeled **Monitor Flow at To End**. This determines which magnitude of flow should be reported. If checked, the flow at the To end of the line will be reported, otherwise the flow magnitude at the From end is used.

Insert

If you came to the Interface Element Dialog by pressing the Insert New Element button on the Interface Information Dialog, only the **Insert**, **Cancel**, and **Help** buttons will be available. Click the Insert button to add the element you have just defined to the list of elements comprising the interface. After you click Insert, the dialog will disappear, and the Interface Elements table on the Interface Information Dialog will contain the element you just added.

Replace, Delete

If you arrived at the Interface Element Dialog by clicking on an element in the Interface Elements Table of the Interface Information Dialog, the **Replace**, **Delete**, **Cancel**, and **Help** buttons will be visible. Click **Replace** to modify the interface element according to your specifications on this dialog. Click **Delete** to remove the element from the interface definition.

Cancel

Click **Cancel** to close the Interface Elements Dialog without saving your changes.

Interface Field Information Dialog

Interface field objects are used to show the different values associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

Interface Name

Case insensitive name of an existing interface (12 characters maximum).

Find...

If you do not know the exact interface name you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Field

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Rotation Angle in Degrees

The rotation angle at which the text field should be displayed.

Anchored

If checked, the interface field is anchored to its associated interface display object.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Type of field to show for the interface record. Interface fields may show the interface's name, its MW Limit, MW Flow, percent loading, or any other interface specific field available by clicking the Select a field option and choosing a field from the drop-down list.

Interface Pie Chart Information Dialog

Interface Pie Chart objects are used to graphically show the percentage flow associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

Interface Name

Case-insensitive name of an existing interface (12 characters maximum).

Size

Size of the pie chart. Note that the pie chart's size and color can be set to change automatically when the interface's loading is above a specified limit. Please see [Online Display Options](#) for details.

Actual Value (read-only)

In Run Mode, this field shows the current MW loading for the interface.

Percent (read-only)

In Run Mode, this field shows the current percentage loading for the interface; if the MVA rating is zero, the percentage is defined as zero, as well.

Anchored

If checked, the interface pie chart object is [anchored](#) to its associated interface.

MW Rating

MW limit for the interface, using the current limit set for the case.

Style

The style of the pie chart. It can be set individually for the particular pie chart, or it can use the style specified in the [Interfaces Pie Chart Options](#).

Automatically Inserting Interfaces in Case

The **Inserting Interfaces Dialog** is used to insert a group of [interfaces](#). Only interfaces between adjacent areas or adjacent zones can be inserted automatically; single-branch interfaces between buses must be inserted with line insertion options. Adjacent areas or zones are those that share at least one tie line. To reach this dialog, first open the Interfaces Display by selecting **Aggregations > Interfaces** from the [Model Explorer](#), and then from the [case information toolbar](#) select **Records > Auto Insert Interfaces**. This dialog is NOT brought up through the menu item **Auto Insert > Interfaces** found on the [Quick Insert](#) ribbon group of the [Draw](#) ribbon tab, which is for inserting [interface objects](#) on an oneline diagram.

The Inserting Interfaces Dialog sports the following fields:

Type of Interfaces to Insert

Select the type of interfaces to insert. Area-to-area interfaces join adjacent areas, while zone-to-zone interfaces join adjacent zones. The name of the new interface defaults to "Area1- Area2" or "Zone1-Zone2" with an **Optional Prefix**.

Optional Prefix

This field allows you to specify an optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name.

Delete Existing Interfaces

If this option is checked, then all existing interfaces are deleted before inserting the new interfaces. By default, this option is checked. If this option is not checked, the existing interfaces are not deleted. However, new interfaces will automatically overwrite any existing interfaces having the same name.

Only Insert Between Areas/Zones with Area/Zone Filters Set

If this option is checked, the set of potential areas or zones for inserting interfaces is limited to those for which the area/zone filter setting is *Yes*.

Limits

Simulator can either calculate an interface rating based on the ratings of the components included in the interface, or the user can specify a set of ratings to be used for the interface. If neither of these options is used to set an interface limit, then by default, the interface limits are left as 0, indicating no limit has been applied.

Insert Interfaces

Click this button to insert the interface records into the case.

Cancel

Closes the dialog without modifying the list of interfaces.

Nomogram Information Dialog

The Nomogram Dialog is used to create, modify, or delete nomogram records in both the Edit and Run Modes. Nomograms are used for combining a pair of [interface objects](#) for the purpose of monitoring a combined flow restriction on the two interfaces together.

This dialog has the following controls:

Nomogram Name

An alphanumeric identifier for the nomogram.

Interface A

The first interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

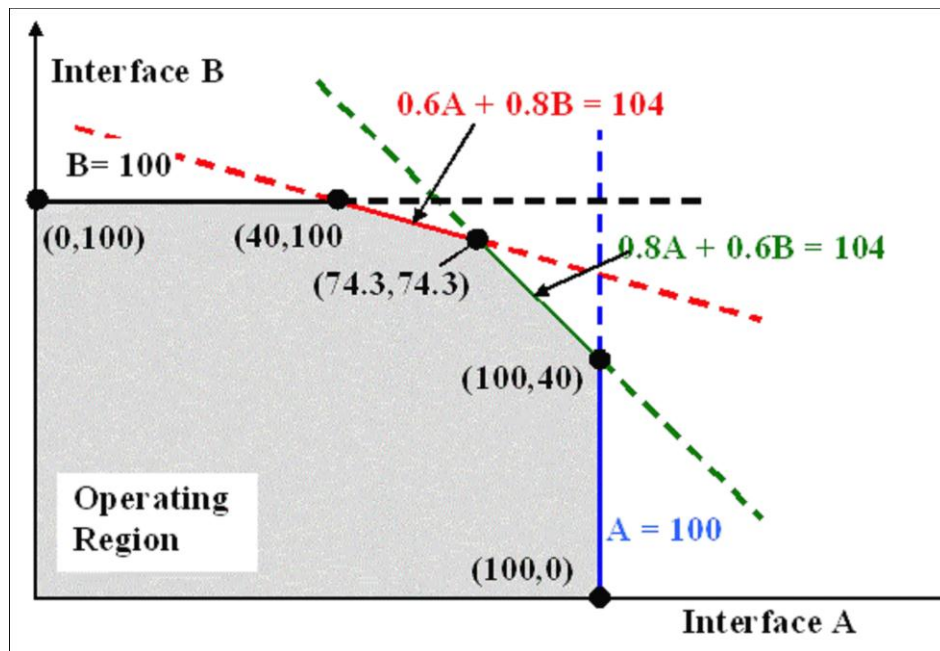
Interface B

The second interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

Nomogram Breakpoints

This section is used for defining the limit boundaries for the interface. The limit boundaries are defined by inserting nomogram breakpoints. These breakpoints correspond to a pair of MW flows on each interface. In other words, you define the amount of flow allowed on Interface B when interface A is at a certain amount. Typically you will have a flow limit on Interface B that is constant for a certain range of flow in interface A. However, at some point as the flow on Interface A increases, the limit of flow on Interface B can start to decrease due to desired flow limit restrictions of the combined interfaces. At some point, the limit of Interface A would reach a maximum amount and remain constant, and the range of flow on interface B would be fairly small due to the heavy loading in Interface A.

To build this Nomogram Limiting Boundary, begin by right-clicking in the Nomogram Breakpoints list and choose Insert Point. Note that the boundary definition must be a convex piecewise linear curve. You would typically begin by defining the flow limit allowed on Interface B when the flow on interface A is small or zero. Then define breakpoints where the limit on B decreases as the flow on A increases. Eventually you will define a point where the flow on A reaches a limit as the flow on B continues to decrease towards zero. In short, the nomogram limiting boundary is actually a combination of boundary limits that are scaled combinations of the individual interface limits. See the image below for an example of a nomogram limiting boundary.



Injection Group Overview

An injection group is a collection of loads, generators, switched shunts, and/or other injection groups. In that respect, injection groups are somewhat analogous to areas and zones. However, unlike with areas and zones, generators, loads, switched shunts, and injection groups can belong to more than one injection group. Moreover, a single injection group may contain generators, loads, and switched shunts from several different areas and zones. Thus, injection groups are useful when modeling a collection of generators, loads, and switched shunts that act together as a unit, regardless of each individual's area or zone affiliation. The most common use for injection groups is to model a transfer of power from one group of generators and loads to another for [PTDF](#) calculations and for [PV/QV analysis](#). They are called **injection groups** because their components (generators, loads, and switched shunts) are objects that inject power into the network.

Creating Injection Groups

Injection groups can be created from the [Injection Group Display](#), [Injection Group Dialog](#), [Generator Display](#), [Load Display](#), [Switched Shunt Display](#), or by selecting a group of display objects on the online display.

To create an injection group from the Injection Group Display, select **Aggregation > Injection Groups** in the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists and provides a summary of all defined injection groups. Click the right mouse button on this display, and select **Insert** from the resulting [local menu](#) and the [Injection Group Dialog](#) will open with a default name assigned to the new injection group. The Injection Group Dialog allows the addition of participation points to the new injection group and allows updating existing injection groups. The **Load** option on the local menu of the Injection Group Display can also be used to import injection groups from an Injection Group Auxiliary Data File and the **Auto Insert Injection Groups** option can be used to automatically insert injection groups based on options set in the [Auto Insertion of Injection Groups Dialog](#).

If the [Injection Group Dialog](#) is already open, clicking the button labeled **New** will create a new injection group. A prompt will then ask for the name of the injection group to add.

To create an injection group from the [Generator Display](#), [Load Display](#), or [Switched Shunt Display](#), select the elements to add to the injection group from one of these displays, right-click on the selection to bring up the local menu, and select **Create Injection Group from Selection** from the local menu. The [Injection Group Dialog](#) will open with the selected elements added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog.

To create an injection from a selection of display objects, select the objects to add to the injection group. Only generators, loads, and switched shunts will be added to an injection group, but if other objects are selected, the injection group will still be created. The objects that cannot be added to an injection group will simply be ignored. Once all objects are selected, right-click on the selection to bring up the local menu. On the local menu select **Create Injection Group from Selection**. The [Injection Group Dialog](#) will open with the selected objects added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog. More than one object must be selected on the display for the create injection group option to be available.

Auto Insert Injection Groups

Many times injection groups need to be created comprised of generators, loads, or switched shunts grouped by areas, zones, super areas, owners, or some other criteria. Injection groups can be created automatically by setting the grouping options by using the Auto Insertion of Injection Groups Dialog.

The Auto Insertion of Injection Groups Dialog can be accessed by selecting **Records > Auto Insert Injection Groups...** from the case information toolbar on an [Injection Group Display](#). The following options are available:

Element Type

This option is used to select the type of element, **Generators**, **Loads**, or **Switched Shunts**, to add to the new injection groups. Only a single type of element can be added to the new injection groups.

Group By

This option dictates the common property by which all elements in the injection group will be grouped. Elements can be grouped by **Areas**, **Zones**, **Super Areas**, **Owners**, or a selected **Custom Field**. If for example the Area option is selected, then injection groups will be created for each of the defined areas in the case. The same holds true for Zones, Super Areas, and Owners. If the **Custom Field** option is selected, then injection groups will be created for all elements whose Custom Field value is the same. For example, if the Custom Field option is selected and Cust Float 1 is set as the field to use, then all of the elements whose Cust Float 1 field are the same will be grouped together. If there are elements whose Cust Float 1 field is set to 1 and there are elements whose Cust Float 1 field is set to 2, then two new injection groups will be created. One will contain all of the elements whose Cust Float 1 field is 1 and the other will contain all of the elements whose Cust Float 1 field is 2. Elements whose selected Custom Field is blank will not be included in any of the new injection groups.

For **Areas**, **Zones**, **Super Areas**, and **Owners**, the creation of injection groups can be limited by only creating injection groups for which the **Selected** field is set to *YES* for these groups. To use this option, check the **Only Selected** checkbox.

Participation Factor

The options for specifying participation factor are the same as those used on the [Add Participation Points Dialog](#).

Delete Existing Injection Groups

When this option is checked, all existing injection groups will be deleted before inserting the new injection groups.

How to Name the Injection Groups

The new injection groups will be named based on the options set in this section. The **Use Prefix** field determines the prefix that will be used on all injection groups. This field can be blank. The **Start at** field determines what integer to start at when counting the new injection groups. The new injection groups will all be numbered in sequential order starting at the value specified. The **Group By** property chosen will also be part of the injection group name. The **Group By** property can be identified based on **Numbers**, **Names**, or **Both**. The field at the bottom of this section provides an example of how the injection groups will be named based on the option settings.

Set Name to Use Prefix

If the options chosen on the injection group dialog are set such that exactly ONE injection group will be created, you can check this option to name that single injection group using the text entered in the **Use Prefix** field. If more than one injection group will be automatically inserted from the

options set in the dialog, this check box option will be ignored, and the injection groups will be create as usual, with the Use Prefix text used as a prefix for the naming convention options selected.

Do Insert Injection Groups

Click this button to implement the options selected and create the injection groups.

Save to Aux

This saves the option settings specified on the dialog to an auxiliary file.

Deleting Injection Groups

To delete an injection group, select **Aggregations > Injection Groups** from the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists all defined injection groups. Click the right mouse button on this display on the injection group to delete, and select **Delete** from the resulting local menu.

Alternatively, if the [Injection Group Dialog](#) is open, the injection group listed in the Name dropdown box can be deleted by clicking the **Delete** button.

Injection Group Display

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

Name

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

Number of Gens

Identifies the number of generators contained in the injection group.

% MW Gen ParFac

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

Number of Loads

Identifies the number of loads contained in the injection group.

% MW Load ParFac

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

% Mvar Load ParFac

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Number of Shunts

Identifies the number of switched shunts contained in the injection group.

% Mvar Shunt ParFac

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

Total MW Injection

Indicates the current total MW injection of all elements in the injection group.

Total Mvar Injection

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

Participation Points Tab

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

Injection Group Dialog

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

Gens

Displays the number of generators contained in the injection group.

Loads

Displays the number of loads contained in the injection group.

Shunts

Displays the number of switched shunts contained in the injection group.

% MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points can be changed by

entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

Import PTI Subsystems Dialog

Injection groups can be created from PTI subsystem definition files from the [Injection Groups Case Information Display](#). Click the right mouse button on this display and select **Load > Subsystem from *.sub file** from the resulting local menu. From the **Import Injection Groups Dialog**, select the file name for the file to import.

If there is no ambiguity in the import file, new injection groups will be created with the defined subsystem names without further prompts to the user. No ambiguity means that either participation points are defined for buses that have either load or generation but not both or that no participation points are defined and that injection group participation points will be defined based on the maximum generation for each on-line generator in the defined subsystem. If participation points are defined without ambiguity, each participation point will be assigned to either the load or generation at the bus split equally across all loads or generators at the bus.

If there is ambiguity in defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed. This dialog prompts the user how participation points should be handled for buses with both load and generation or buses with no load or generation.

Buses with Load and Generation

This option allows the user to select how a defined [participation point](#) will be assigned if the participation point bus has both load and generation.

Assign Participation Point to Generation

The participation point will be assigned to generation at the bus. The participation point will be split equally across all generators, either on-line or off-line, at the bus.

Assign Participation Point to Load

The participation point will be assigned to load at the bus. The participation point will be split equally across all loads, either connected or not, at the bus.

Buses with No Load or Generation

This option allows the user to select how a defined participation point will be assigned if the participation point bus has no load or generation.

Ignore Participation Point

The participation point will be ignored.

Add Equivalent Load (closed load with ID='99' and 0 MW and 0 Mvar) and Assign Participation Point to this Load

This option adds a connected load at the participation point bus with ID='99' and 0 MW and 0 Mvar. The participation point is then assigned to this new load.

For each subsystem that is read that is found to have ambiguity in the defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed unless the user selects Same Options for All Subsystems (Do not prompt again).

Participation Points Overview

A participation point is a member of an injection group. It is a generator, load, switched shunt, or another injection group that participates in, or contributes to, the output of an injection group. Each participation point record identifies the generator, load, switched shunt, or injection group that fills this role, its participation factor, and how that participation factor is calculated. A participation point's participation factor identifies the degree to which the point will contribute to its injection group's output relative to the other points making up the group. Participation factors may be defined as having a fixed value, or they may be re-calculated with every use to stay true to how they were originally defined. For generators, participation factors may be defined as having a fixed value or being equal to the generator's MW reserve in the direction of increasing output (MAX GEN INC), MW reserve in the direction of decreasing output (MAX GEN DEC), or their maximum output (MAX GEN MW).

Each time that an injection group is used, the participation factors for all participation points in the group are normalized so that the factors of all participating participation points sum to 100%. The normalized factors dictate how much a generator, load, or switched shunt contributes to the output of an injection group. The normalization accounts for the re-calculation of the participation factors with each use of the injection groups and the exclusion of participation points because of options set with the various tools that use injection groups. Participation points can be excluded for various reasons including, but not limited to, the exclusion of generator participation points because the generator is not on AGC control or because the generator is at a minimum or maximum limit.

Participation points are added to or deleted from an injection group using the [Injection Group Dialog](#). The Injection Group Dialog houses the [Participation Point Records Display](#), which allows the addition or deletion of points and opening the [Participation Points Dialog](#) to obtain more information about the points.

Participation Point Records Display

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

Point Type

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

Number

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

Name

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

ID

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

AutoCalc?

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

Initial Value

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

SPECIFIED The participation factor was specified as a constant.

MAX GEN INC The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

MAX GEN DEC The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

MAX GEN MW The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

SPECIFIED The participation factor was specified as a constant.

LOAD MW The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

SPECIFIED The participation factor was specified as a constant.

MAX SHUNT INC The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

MAX SHUNT DEC The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

MAX SHUNT MVAR The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

SPECIFIED The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

Injection Group 1 Injection Group 2

Gen 1, ParFac = 50 Gen 5, ParFac = 40

Gen 2, ParFac = 30 Load 1, ParFac = 20

InjectionGroup 2, ParFac = 20 Load 2, ParFac = 20

Total = 100 Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:

Gen 1, Normalized ParFac = $50/100 = 0.5$

Gen 2, Normalized ParFac = $30/100 = 0.3$

Gen 5, Normalized ParFac = $20/100 * 40/80 = 0.1$

Load 1, Normalized ParFac = $20/100 * 20/80 = 0.05$

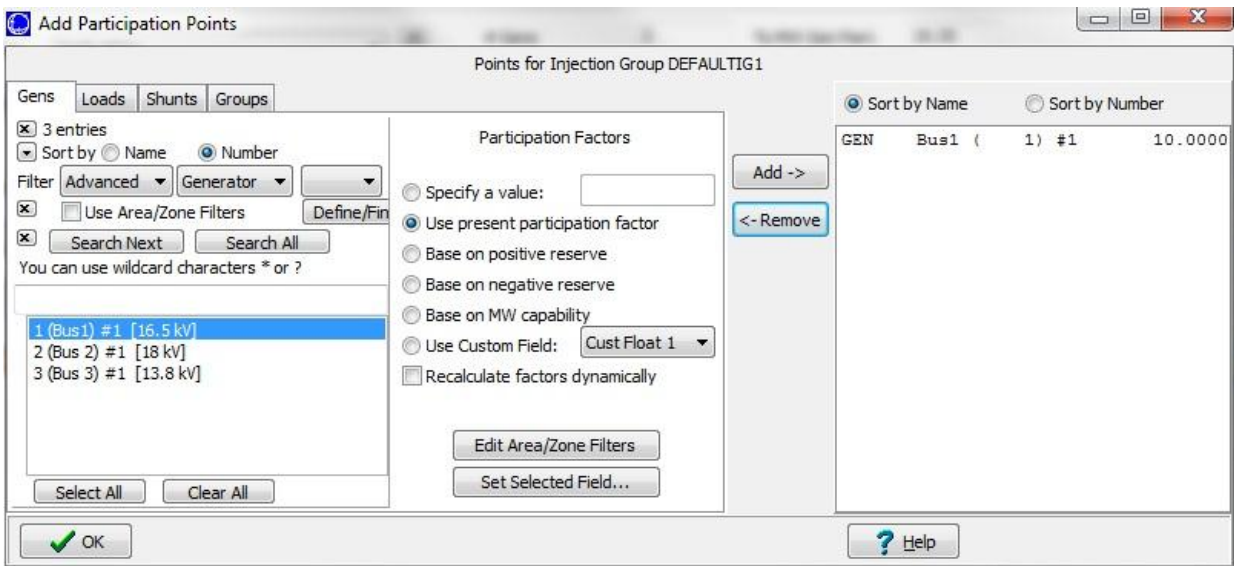
Load 2, Normalized ParFac = $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

Add Participation Points Dialog

The Add Participation Points Dialog enables the addition of participation points to an [injection group](#) as well as the modification of existing participation points. This dialog is accessed from a [Participation Point Records Display](#) by right-clicking in the participation points list and selecting **Insert** from the resulting [local menu](#). Participation Point Records Displays are available from the [Injection Group Dialog](#) or [Injection Group Display](#).



The Add Participation Points Dialog features four tabs, one for adding generator points, one for adding load points, another for switched shunts, and a fourth for adding points from injection groups that have already been defined.

The four tabs are almost identical and contain the following controls:

Filtering

If the **Use Area/Zone Filters** box is checked, the list box beneath it, which lists generators, loads, switched shunts or previously defined injection groups depending on the tab, will list only those elements contained in areas or zones or by owners whose [area/zone/owner filter](#) setting is **YES**. If this box is not checked, all generators, loads, and switched shunts in the case will be listed. Injection groups will not be filtered using the area/zone/owner filter.

Alternatively, a custom filter can also be defined by clicking on the **Define Filter** button. This will open the [Advanced Filter](#) dialog, which allows the customization of a filter for determining the devices to list in the display.

Once the list of devices has been set, with or without filtering, the list can be searched using the advanced search techniques in Simulator. These techniques allow the list to be sorted by name or by number and allow the use of wildcard characters. Simply choose Name or Number, and type the name or number of interest in the box. Simulator will look for and highlight the first matching device in the list. If the first device is not the one of interest, use **Search Next** to find the next device that matches the search criteria. For a comprehensive list of all objects matching the search criteria, press **Search All**.

Element List

The box that occupies the left side of each tab lists the generators, loads, switched shunts, or injection groups (depending on which tab is active) that can be added to the injection group. Injection groups will only show up in the element list if they do not cause a circular reference by being added to the current injection group. This means that any injection group that would cause the current injection group to link back to itself will be omitted from the list. Multiple elements can be selected from each of these lists. To select several elements in a row, drag the mouse to highlight the elements to be added. Alternatively, click the first element to add, press and hold the shift key, and click the last element to add. To select elements that are not adjacent in the list, click the first element you to add and hold down the CTRL key while clicking the other elements to add.

Once an element is selected in the Element List, it is ready to be added to the injection group.

Participation Factors

There are several options for defining the participation factors of the selected points.

For generators, a value can be specified, the generator's present participation factor (which comes from the case and is displayed in the [Generator Display](#)) can be used, the participation factor can be calculated as the difference between its present output and either the unit maximum or minimum, its maximum output capability can be used, or the value contained in a selected Custom Field can be used.

For loads, a value can be specified, the participation factor can be based on the load's size, or the value contained in a selected Custom Field can be used.

For switched shunts, a value can be specified, the value contained in a selected Custom Field can be used, or the factor can be based on positive reserve, negative reserve, or MVAR capability.

For injection groups, a value can be specified to use for every point in the injection group, the values already defined for the injection group can be used, or the value contained in a selected Custom Field can be used. These three options are available if the option to **Include Individual Group Points** is checked. When this option is checked, copies of the participation points from the selected injection groups are made and added to the current injection group with the participation factors based on the selected option. When **Include Individual Group Points** is not checked, entire injection groups are included, indicated by the Point Type of *INJECTIONGROUP* in the [Participation Point Records Display](#), in the current injection group and the only option for adding the points is to specify the value for the participation factor. When including an injection group in another injection group as the entire group, the normalization and use of the participation factors follows the discussion of ParFac found in the [Participation Point Records Display](#). Including an injection group in another injection group as the entire group is useful when the injection of the entire group needs to be changed relative to other individual elements or other injection groups.

Recalculate Factors Dynamically

If this box is checked, the participation factors of the points being added will be automatically updated every time the points are used. Such points will then have an **AutoCalc** value of *YES*. If this box is not checked, the participation factors of the points being added will be fixed at the values defined at the time they were created.

To add the points that have been selected, click the **Add ->** button. The new points will be added to the list box on the right.

To update existing points, follow the same process for adding points and simply set the participation factors with the new values. When the **Add->** button is clicked, the existing points will be updated with the new values.

The list box that occupies the right side of each tab lists the points that already comprise the injection group. To delete specific points from the injection group, select them from this list box and click the **<-Remove** button.

To close this dialog, click **OK**.

Transaction Dialog

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

Information

Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

Transaction To Area

This is the "to" area for the transaction.

Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

Determine Price in OPF

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

Piecewise Linear Transaction Cost Curve

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

Custom

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.

Bus View Display

The **Bus View Display** feature provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus. The bus view displays enable convenient bus-by-bus navigation through the power system. From the bus view, you can find out a bus' voltage and angle, the load, shunt compensation, and generation connected to the bus, and the flows on all lines emanating from the bus. You can also discover the bus' area and zone affiliations, as well as the bus' marginal cost. Moreover you can find out all information about the elements associated with the bus by directly invoking their associated information dialogs. The advantage of the bus view displays, is that they are auto-created oneline diagrams.

Along the top of the bus view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of buses you have viewed thus far. To move the bus view to a particular bus, type a number or name into the edit box next to the caption **Bus**. If a number is typed the case will be scanned for a bus of that number, otherwise the case will be scanned for a bus that starts with the string entered. If you're unsure which bus that you are looking for click the **Find...** button to bring up a display that allows you to search using wildcard strings for the bus. The **History** menu will show a list of the last 30 buses which have been visited in a bus view. There are also drop-downs in this top panel for **Options** and **Views**. These are discussed below.

Below this top panel sits the actual bus display. The bus you have chosen to inspect, which we shall refer to as the *target bus*, is represented by a long, thick horizontal line. Notice that the bus' voltage in kV and per unit, its angle, and its marginal cost are specified to the left of the bus. Any loads and generating loads connected to the target bus are drawn above the bus symbol, along with their associated annotation. Also any radial connections to a single bus containing load, generator or switched shunts will be drawn above the bus symbol.

Emanating from the bottom of the bus symbol are all transmission lines and transformers that connect the target bus to its neighbors. The transmission line and transformer symbols are equipped with pie charts and annotation identifying flows as measured at the target bus, as well as arrows to identify the direction of MW flow on the branch. Neighboring buses are represented as filled rectangular regions, with symbol indicators included if other types of devices, such as loads, generators, etc., are attached. When you drag the mouse over one of the neighboring bus symbols, it turns into a pointing finger. Clicking the left mouse button when the mouse cursor is in this shape redefines the target bus to be the bus whose symbol you just clicked. The bus view display is redrawn to show the same sort of display for the newly chosen target. You can go back to the previously displayed target bus by clicking the **Back** arrow, and then return to this record by clicking the **Ahead** arrow.

It is useful to think of the bus view displays as nothing more than an additional oneline diagram. In other words, you interact with the objects drawn on the bus view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that object's local menu, which includes a link to the object's associated information dialog. As on a conventional oneline diagram, flows on a bus view display can be animated. Right-clicking on the bus view display's background will generate the same local menu as other oneline diagrams.

The bus view display can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group
- Go to the [Onelines](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group.
- Right click on the bus of interest on the oneline diagram to display the bus' local menu, and choose Bus View. The bus view display will open with the selected bus already displayed.

- From any of the case information displays that convey bus information, right click on a record to bring up its local menu, and choose Bus View Online. The bus view display will open with the corresponding bus already displayed.
- Click the **Bus View** Button on the [Case Information Toolbar](#).

To switch between the bus view and the main oneline, use the **Open Windows Menu** available on the **Views** ribbon group on the [Case Information](#) ribbon tab, [Case Information](#) ribbon tab, or [Windows](#) ribbon tab. To close the bus view display, simply close the form using the **X** button in the top right corner of the bus view display.

Note that when viewing a consolidated superbuss while using consolidation via the topology processing tool, Simulator will always show circuit breaker symbols on branches, loads, generators, and switched shunts which are open (out of service.)

Bus View Options

Layout Options

Show Serial Buses

When putting in the branch connections, the Bus View display will look out into the network and find the next bus which has more than two neighbors. It will then make this the destination bus for that branch of the Bus View. The intermediate buses will then be shown in order above the destination branch. This option works especially well in systems with a lot of multi-section lines.

Show Equivalent Lines

This option indicates to the Bus View display whether or not to include branches representing equivalent circuits as connections in the display.

Represent Multi-Section Line Objects

This option indicates to the Bus View display whether or not to draw multi-section line display objects when appropriate. If this option is not checked, then the intermediate buses of the multi-section lines will be rendered normally. If this option is checked then the intermediate buses of multi-section lines will not be rendered on the bus view and only the terminals of the multi-section line will appear.

Show Field Suffixes

This option specifies whether to display the field values units as suffixes. If this option is not selected, all the fields will be display as a value without units.

Include Field Labels

Selecting this option will place labels for each displayed field on the bus view diagram.

Default Drawing Values

Choosing this option will open the Default Drawing Values for New Objects dialog. Changing these options can change some of the drawing aspects of the bus view, including device color and font size or color.

Dynamic Formatting

Click on this to bring up the [Dynamic Formatting](#) for case information displays, and all views and onelines.

Number of Tiers

The bus view can display one or two "tiers" of buses in the display. Use this selection from the **Options** menu to toggle the number of tiers displayed.

Open Multiple Bus Views

This option indicates whether to open multiple bus views simultaneously. Choices are never, always, and prompt for confirmation when a new additional bus view is about to open.

Show Hints

When this option is checked, holding the cursor over an object will briefly pop up a hint box containing information about that object.

Views

Define Custom View

The fields displayed on the bus view can be customized using this option. Clicking on this option will open a customization settings display, in which you can add and remove field definitions for the objects on the bus view display. Customized bus view layouts can be saved with the case for recall, identifiable by a custom bus view layout name. Custom layouts can also be saved to a file for loading into another load flow case.

Input Data

Switching the bus view to Input Data changes the bus view from displaying system state information to displaying input data information. For example, switching to Input Data view will display line impedances and limits, generator minimum and maximum outputs, etc. The default Input Data view fields can be modified using the Define Custom View customization dialog.

System State

Switching the bus view to System State changes the bus view from displaying Input Data information to displaying system state information. This will result in line flows being displayed, voltage and angles displayed, etc., of the current solution state of the system. The default System State view fields can be modified using the Define Custom View customization dialog.

Substation View Display

The Substation View Display feature is analogous to the [Bus View Display](#).

Along the top of the substation view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of substations you have viewed thus far. Substation View options are available from the **Options** menu. To find a substation using search tools, click on the **Search For** button. The next two controls following the Substation label allow to specify a substation name (in the first text box) or a number (in the second text box). If you type a number or name that does not exist, the substation display will continue displaying the current substation.

Below this top panel sits the actual substation view display.

Just as with the bus view, it is useful to think of the substation view displays as nothing more than a oneline diagram. In other words, you interact with the objects drawn on the substation view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that objects local menu, which includes a link to the object's associated [information dialog](#). As on a conventional oneline diagram, flows on a substation view display can be animated. Right-clicking on the substation view display's background will generate the same [local menu](#) as other oneline diagrams. The substation view can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Substation View** from the Views Ribbon Group
- Go to the [Onelines](#) ribbon tab, and click on the **Substation View** from the **Views** ribbon group.
- Right click on the substation of interest on the oneline diagram to display the substation's local menu, and choose Substation View. The substation view display will open with the selected substation already displayed.
- From any of the case information displays that convey substation information, right click on a record to bring up its local menu, and choose Substation View Oneline from the Substation Records menu. The substation view display will open with the corresponding substation already displayed.
- Click the **Substation View** Button under the **Records** menu on the [Case Information Toolbar](#).

To switch between the substation view and the main oneline, use the **Window** menu tree on the main menu. To close the substation view display, simply close the form using the **X** button in the top right corner of the substation view display.

Using Labels for Identification

Most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) may have an alternative names assigned to them. These alternative names are called labels. Labels allow you to refer to equipment in the model in a way that may be unique to your organization. Labels may thus help clarify which elements are described by a particular set of data, especially when the short names employed by the power system model prove cryptic.

Furthermore, since labels are likely to change less frequently than bus numbers, and since a label must, by definition, identify only one power system component, they may function as an immutable key for importing data from [auxiliary files](#) into different cases, even when bus numbering schemes change between the cases. Labels must be unique for devices of the same type, but the same label can be used for a device of a different type.

Information dialogs corresponding to buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces feature a button called **Labels**. If you press this button, the device's Label Manager Dialog will appear. The [Label Manager Dialog](#) lists the labels associated with the device and allows adding, deleting, and modifying labels. The **Labels (All)** field (variablename = LabelsAll) in a [Case Information Display](#) lists all of the labels that have been associated with a device. The device's primary label is the one that is listed first in this field. This field lists all labels assigned to a device as a comma-delimited string. Even though multiple labels may be assigned to a device, any label can be used to import data from [auxiliary data files](#). The Label Manager Dialog can be used to assign labels or the string of labels can be entered directly in the **Labels (All)** field.

Labels can be used to map data from an auxiliary data file to a power system device. Recall that [auxiliary data files](#) require you to include a device's key fields in each data record so that data may be mapped to the device. Labels provide an alternative key. Instead of supplying the bus number to identify a bus, for example, you can supply one of the bus's labels. The label will enable Simulator to associate the data with the device associated with that label. This mechanism performs most efficiently when the primary label is used, but other labels will also provide the mapping mechanism. The **Label (for use in input from AUX or Paste)** field (variablename = Label) is used for importing data using labels and is blank when viewing in a case information display. Keep in mind that all devices read via an auxiliary file using the label field should have a non-blank label. Otherwise, information for that device will not be read. Even if the primary or secondary key fields are provided with the device, as long as the label field is present, that is the only field that will be used to identify the device. New devices cannot be created by simply identifying them by label. Either the primary or secondary key fields must be present to create a new device and the label field should not be present.

Again, it is important to remember this: a single power system device may have multiple labels, but each label may be associated with only one device of a particular type. This is the key to enabling data to be imported from an auxiliary file using labels.

Saving Auxiliary Files Using Labels

All devices that can be identified by labels will have the **Labels (All)** and **Label (for use in input from AUX or Paste)** fields available in their case information displays. In order to save auxiliary files that identify devices by label, the two label fields should be added to the case information display prior to saving the data in an auxiliary file. Because the **Label (for use in input from AUX or Paste)** field will be blank when saved in the auxiliary file, this field must be populated with one of the labels in the **Labels (All)** field before loading the auxiliary file back in. Keep in mind that devices with blank labels cannot be identified when loading in an auxiliary file, so avoid saving auxiliary files by label if all devices do not have labels.

Many devices require [SUBDATA](#) sections. These sections have custom formats specific to the type of information that they contain. When saving auxiliary files with devices that require SUBDATA sections, the user can choose to use primary or secondary key fields or labels to identify devices in the SUBDATA sections. The user will either be prompted when saving the devices, or there is an option to change the key field to use when saving subdata sections on the PowerWorld Simulator Options dialog under the [Case Information Displays](#) category. When choosing to use labels, if a device has a label, it will be used. If it is a device that can be identified by buses and bus labels exist, bus labels will be used. Finally, if the device does not have a label and the buses do not have labels, the primary key for the device will be used for identification.

Devices that have SUBDATA sections that contain other devices that can be identified by labels include: contingencies, interfaces, injection groups, post power flow solution actions, and owners.

The setting to choose which identifier to use for the SUBDATA sections does not just apply to SUBDATA sections. Often when saving groups of options, this setting will apply to everything being saved with those options and not just the SUBDATA sections. This includes contingency options, ATC options, limit monitoring settings, and PVQV options. In these cases, there will be a prompt asking the user to decide which identifier to use in the auxiliary file.

Loading Auxiliary Files SUBDATA Sections Using Labels

The various SUBDATA sections that represent references to other objects can also be read using labels. Examples include contingencies, interfaces, injection groups, post power flow solution actions, and owners. When reading a SUBDATA section such as this, PowerWorld makes no assumption ahead of time about what identification was used to write this SUBDATA section. Instead, an order of precedence for the identification is as follows

	Identification	Explanation	Example
1	Key Fields	assumes that the strings represent Key Fields	BRANCH 8 9 1
2	Secondary Key Fields	assumes that the strings represent Secondary Key Fields	BRANCH Eight_138 Nine_230 1
3	Labels for component objects	the key/secondary key fields for some objects consist of references to other objects. An example of this is the BRANCH object that is described by the From Bus, To Bus, and Circuit ID. This assumes that labels of the component objects are used.	BRANCH Label8 Label9 1
4	Labels	Assumes that the string represents one of the Labels of the object	BRANCH LabelForBranch

Special Use of Labels

There are a few special cases where objects have fields that identify other devices. These devices can be identified by label but not in the conventional means because the label field applies to the object that contains the device and a SUBDATA section is not necessary. These special cases include: (Note all fields given below are by variable name because the use of labels is most relevant with auxiliary files.)

ATC Scenarios

[ATC Scenario](#) change records usually contain primary key fields to identify the devices that should be adjusted during the scenario. If using labels, these primary key fields will be replaced with a single **Label** field. The use of this field is different because the Label field refers to the device in the change record and not to the change record itself. When labels are used with ATC scenarios, device labels only can be used. Bus labels cannot be used to identify devices for which no label exists but a bus label does.

ATC Scenarios are saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

ATC Extra Monitors

[ATC Extra Monitors](#) identify either branches or interfaces to monitor during the ATC analysis. These devices are identified in the **WhoAmI** field of ATC Extra Monitor records. Usually, the **WhoAmI** field is a special format that contains key field tags. Optionally, this field can use the label of the device for the extra monitor. If the device label is not available, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

ATC Extra Monitors area saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

Model Conditions

Devices in [Model Conditions](#) are usually identified by the **WhoAmI** field which is in a special format that contains key field tags. Optionally, this field can use the label of the device. If the device label does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Model Expressions

[Model Expressions](#) contain Model Fields. Model Fields are identified by the **WhoAmI** fields in the Model Expressions. Usually, the WhoAmI fields are in a special format that contains key field tags. Optionally, these fields can use the label of the device associated with the Model Field. If the device does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

Bus Load Throw Over Records

[Bus Load Throw Over Records](#) are used with contingency analysis. These records have an option to identify the bus to which the load will be transferred by either number or name_kV combination. If choosing to identify objects by label, the **BusName_NomVolt:1** field will contain the label of the bus instead of the name_kV combination.

Bus Load Throw Over Records will be saved in an auxiliary file if choosing to [Save](#) settings on the Contingency Analysis dialog.

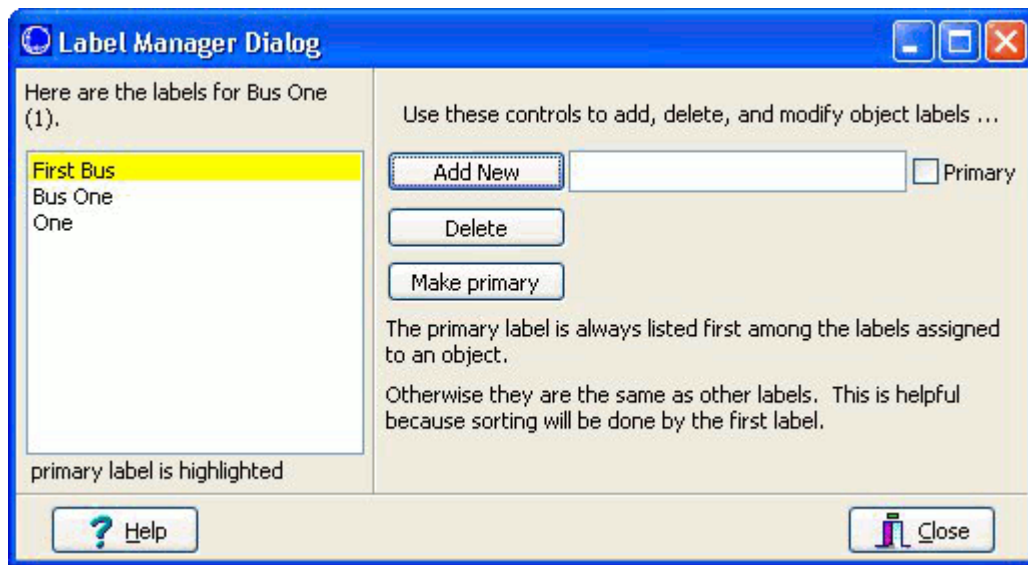
Injection Group Participation Points

All participation points and the injection groups to which they belong can be listed on the [Injection Group Display](#). Load, generator, and shunt devices that can be assigned to a participation point must be identified by bus and ID. The bus can be identified by either the number or name. When identifying by name, the **BusName_NomVolt** field is used to provide the name_kV combination for the bus. If choosing to identify devices by label, this field instead will contain the label of the device. If the device does not have a label but the bus does, the bus label will be used instead in conjunction with the ID of the device. Even if the device does contain a label, the ID field must be included in any auxiliary file that is going to be loaded because it is a key field. Injection groups can be included in other injection groups. Injection groups can be identified by label, even though this is not a normal thing to do. If any injection groups have labels and these injection groups are included in other injection groups, their labels will also appear in the BusName_NomVolt field. If they do not have labels, they will be identified by the injection group name that appears in the PPntID field.

Label Manager Dialog

The Label Manager Dialog allows the creation and modification of [labels](#) to identify power system devices. Labels can be used for most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) as alternate names for these devices. See the [Labels](#) topic for more information on how labels can be used.

The Label Manager Dialog can be accessed by clicking the Labels button on any of the information displays for objects that allow labels.



List of Labels

Multiple labels can be assigned to a single object. The list on the left of the dialog shows all of the labels that have been assigned with the primary label listed first and highlighted in yellow.

Add New

Click this button to add a new label after entering the new label in the box to the right of the button. If this new label should be the primary label, check the **Primary** box. Labels must be unique for each type of object. That means that there cannot be two buses labeled *One*. However, there could be objects of different types with the same label. There could be a bus and a generator both labeled *One*. If attempting to label an object with a label that has already been used by the same object type, the label will be assigned to the current object and removed from the object to which it was previously assigned.

Delete

Click this button to delete a label after first selecting the label in the list of labels.

Make Primary

Click this button to make a label the primary label after first selecting the label in the list of labels. The primary label will be moved to the top of the list. The primary label will always appear at the top of the list and will be used for sorting when sorting by label.

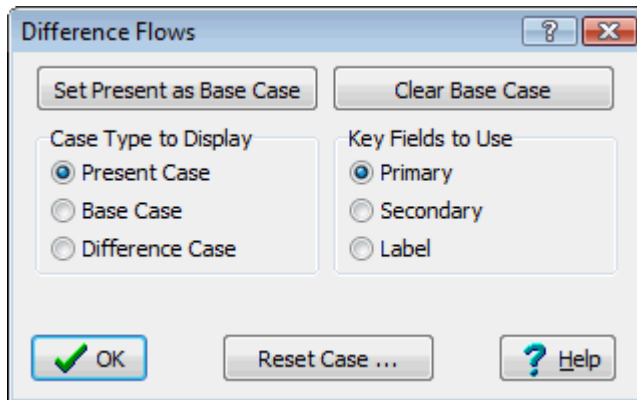
Difference Flows

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, Difference Flows can be used to show the difference in transmission line flows and bus voltages resulting from a contingency or a change in power transfer between two areas. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

For more information about how to use Difference Flows see [Using Difference Flows](#).

The Difference Flows Dialog can be accessed by:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows > Difference Flows** from the [Other Tools Ribbon Group](#).
- Selecting **Difference Flows** from the [online local menu](#).
- All Difference Flows Actions are controlled via the Difference Flows Dialog or the drop-down list on the Difference Flows toolbar button.



Case Type to Display

Use of this feature affects all aspects of the Simulator environment. When using the Difference Flows tool, information shown on online diagrams, case information displays, and [power flow lists](#) is governed by which of the three [Difference Flow Case Types](#) (base, present or difference) is currently being displayed.

Key Field to Use

The difference flows feature must make a comparison between the Present Case and the Base Case to determine which objects between these two case match one another. The Key Fields to Use specifies whether this matching is done using Primary Keys (bus numbers), Secondary Keys (bus names/Nom kV pair), or by Label.

Difference Flows: Case Types

When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three case types is currently being displayed. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

The case types available are:

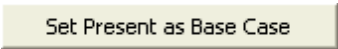
- **Base Case** – A solved power system that serves as the reference for the difference flows tool. To establish a base case, set up a solved power system corresponding to the desired operating point. Open the Difference Flows Dialog and click the button labeled **Set Present as Base Case**.
- **Present Case** – The operating point used in the Difference Flows comparison. **Note:** The Present Case must have the same numbering scheme as the Base Case for proper operation of the Difference Flows tool. See [Topological Differences](#) for more information.
- **Difference Case** – The difference between the Present Case and the Base Case values. The values displayed in the Difference Case are established using the Base Case as the reference. When showing status fields for branches, generators, etc... fields will show up as either OPEN/CLOSED or CLOSED/OPEN if the status changed. The status in the present case will be listed first. A similar format will be used for fields such as generator AGC Status and other text fields.

To toggle between the different case views, open the [Difference Flows Dialog](#) and select the desired case type. Alternatively, you can click the **Difference Flows** button in the [Other Tools Ribbon Group](#) on the [Tools](#) ribbon tab and select the desired case type.

Using Difference Flows

To use the Difference Flows tool:

- Set up a solved power system corresponding to a desired operating point. This operating point will be defined as the [Base Case](#).
- Select **Difference Flows** from the Run Mode ribbon group on the [Tools](#) ribbon tab, or select **Difference Flows** from the [online local menu](#) in Run Mode.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the Base Case.



- Define the operating point ([Present Case](#)) for which to perform the difference flows comparison. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case selecting **Open Case** from the [Application Button](#). In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- See [Difference Flow Case Types](#) for information on toggling between case views. The currently displayed case type is shown in the [PowerWorld Simulator Status Bar](#). When viewing either the Base Case or Difference Case, the corresponding status bar display will be highlighted.

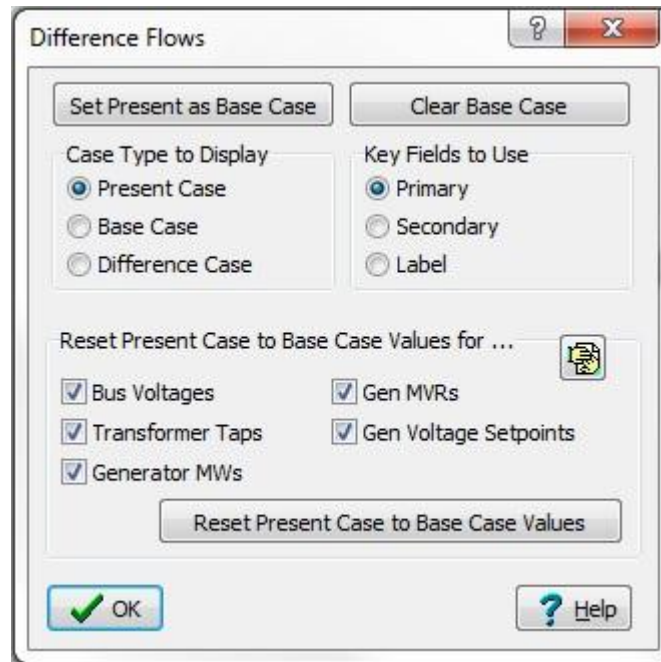


Note that the Difference Flows tool can only be used in Run Mode and the status bar will not display a case type while in Edit Mode.

When showing the Difference Case, most of the fields shown on the onelines and case information displays show the difference between the present value and its Base Case value. For example, on the Generator Records Display, an entry of 0.0 in the MW field indicates that the real power output of the generator did not change. An entry of 10.0 in the MW field indicates that the present real power output of the generator is 10 MW greater than it was in the Base Case.

At any time during a simulation, you can set the present case as the Base Case by clicking the corresponding button on the Difference Flows Dialog.

Conversely, if you have made changes to the present case, and you wish to revert to some or all of the base case values, you can click on the **Reset Case...** button. When you click this button, the dialog will expand to show you options for resetting specific types of values to their base case values.



Expanded Difference Flows Dialog

Once you have indicated which types of values you want to reset (by default, all are selected), you can click on the **Reset Present Case to Base Case Values** button to complete the process of reverting to base case values. Note: once you have finished resetting to the base case values, you can hide the Reset options by clicking the Hide Reset Button in the upper right hand corner of the Reset options panel.

The oneline diagrams and case information displays cannot indicate structural differences in the case very well, such as the addition or removal of a device. To identify such differences, make use of the [Present Topological Differences from Base Case](#) tool to identify topology differences.

Present Topological Differences from Base Case

The Present Topological Differences from Base Case option provides the users a way to compare the topological differences between two difference cases in Simulator.

To compare two cases topologically, you must load the first case you wish to use as the reference into Simulator. Use the [Difference Flows](#) tool to set the case as the base case in memory. Once the reference case has been stored as the base case, open the second or comparison case into Simulator. Now if you check the [Tools](#) ribbon tab, and choose the **Difference Flows Menu** on the [Other Tools Ribbon Group](#), you will see the option **Present Topological Differences from Base Case**. Choosing this option will open the dialog to display the topological differences.

Present Case Topological Differences from the Base Case

Summary | Elements Added | Elements Removed | Elements In Both | Create Bus Swap List

Below is a summary of the comparison between the present case and the base case saved from the Difference Flows Dialog.

Element Type	New	Removed	Both
Bus	0	0	9
Load	0	0	3
Switched Shunt	0	0	1
Generator	0	0	3
Branch	0	0	9
DC Line	0	0	0
Area	0	0	1
Zone	0	0	1
SuperArea	0	0	0
Transformer	0	0	3

☒ Assume base case Areas/Zones which are not in present case meet the Area/Zone Filters

Save and Send Option

All Lists

☐ Use Area/Zone Filters when saving to Auxiliary File

Send To Excel | Save to Text File | Save To EPC File | Save To Aux File | Load Aux File | Close

Topological Differences Dialog

The Topological Differences Dialog contains five tabs. All but the Summary tab are instances of Case Information Displays and thus exhibit characteristics and controls similar to other displays.

Summary

A listing of the types of objects in the case. The first column displays the number of NEW devices in the comparison case that were not in the base case. Alternately, the Removed column displays the number of devices in the base case that do NOT exist in the comparison case. The third column simply displays the number of items that were matched between the two cases.

Elements Added

Tabular listings of all objects that exist in the comparison case, but not in the reference case.

Elements Removed

Tabular listings of all objects that exist in the reference case, but are not present in the comparison case. A right-click option from the pop-up menu on the Elements Removed tables allows for saving the removed elements selected to a GE EPC file format.

Elements in Both

Tabular listings of all objects present in both cases.

Create Bus Swap List

Used for setting up a bus renumbering list. It is possible that discrepancies in topology between the comparison case and the reference case can be due to a difference in bus numbering between the two cases. [Renumbering the buses](#) in the comparison case may take care of most of the topology discrepancies that are reported in this instance.

Data Objects and Fields

The Save and Send Options on the topological difference dialog makes it possible to save Elements Added, Elements Removed, and Elements In Both to file. In addition to these options, special objects and an object field are available to make accessing these objects easier using script actions and will allow the user to customize the fields stored in the file. When the removed elements are saved in the GE EPC file format the elements are saved with a status value of -4.

To provide easier access to the objects that have been identified as being removed when viewing the topological differences, special object types have been created that allow these objects to be accessed via script commands. These objects provide access to the Elements Removed. All of the object names start with *Removed* with the rest of the name just being the usual object name. To identify buses use the object name *RemovedBus*. To get generators use *RemovedGen*, etc. These objects can be accessed through the [SaveData](#) script command and [GetParametersMultipleElement](#) through SimAuto. These object types can only be stored to an auxiliary file. They cannot be read back in, but, the file headers and object types can be modified to make use of the data or the data can simply be used as documentation of case differences.

A field is available for all object types that can be used with the difference flows tool. This field is found under **Difference Flow\In Diff. Flows Base Case?** and indicated whether or not an object was in the difference flows base case. This field is useful for filtering only the new elements that are only in the present case, field value = NO, or all of the elements in both the present and base case, field value = YES. This will make it possible to use script actions or filters to return only the Elements Added or Elements In Both.

Auxiliary Files

PowerWorld has incorporated the ability to import data from data sources other than power flow models into PowerWorld Simulator. Simulator has always had the ability to import supplemental data into a power flow model, but this facility was limited to ten or twelve native data formats targeted to specific applications for which the data had to have a precise format. The extensions described in this help document provide a more versatile interface to data from non-power-flow sources.

In addition to developing auxiliary data formats, a script language was also developed. The script language and auxiliary data formats are incorporated together. This format is described in this help document.

Script/Data files are called *data auxiliary* files in Simulator and typically have the file extension .AUX. These files mostly contain information about power system elements and options for running the various tools within Simulator. They do not contain any information about individual display objects contained on a one-line diagram. There are separate files called *display auxiliary* files that are available for importing/exporting display data from/to Simulator in a text format. These files are distinguished from the *data auxiliary* files by using the extension .AXD. The format for these two types of files is similar, but different object types are supported by each and require that the files be read separately. Currently, there are no script commands that are available for *display auxiliary* files.

Both file types will be generically referred to as auxiliary files. An auxiliary file may be comprised of one or more DATA or SCRIPT sections. A DATA section provides specific data for a specific type of object. A SCRIPT section provides a list of script actions for Simulator to perform.

These sections have the following format:

SCRIPT ScriptName1

```
{  
script_statement_1  
.  
script_statement_n  
}
```

DATA DataName1(object_type, [list_of_fields], file_type_specifier,create_if_not_found)

```
{  
data_list_1  
.  
data_list_n  
}
```

DATA DataName2(object_type, [list_of_fields], file_type_specifier,create_if_not_found)

```
{  
data_list_1  
.  
data_list_n
```

```
}
```

SCRIPT ScriptName2

```
{  
script_statement_1  
.  
script_statement_n  
}
```

Note that the keywords SCRIPT or DATA must occur at the start of a text file line. Auxiliary files may contain more than one DATA or SCRIPT section. These sections always begin with the keyword DATA or SCRIPT. DATA sections are followed by an argument list enclosed in (). The actual data or script commands are then contained within curly braces { }.

Script Command Execution Dialog

The Script Command Execution dialog provides a location for the user to enter [script commands](#) manually, or to load [auxiliary files](#) containing [Script](#) and/or [Data](#) sections previously defined. This dialog is opened by pressing the **Script** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

Loading Auxiliary Files

The first feature of the Script Command Execution dialog is that it provides a location to load previously defined [auxiliary files](#) for the currently loaded case. The **Auxiliary File** menu provides a location for loading an auxiliary file, or simply validating that the [Script](#) and/or [Data](#) sections of an Auxiliary file are correctly formatted in the file.

The **Quick Aux** option allows you to open the [Quick Auxiliary Files](#) dialog for creating a list of auxiliary files to be opened and processed en masse.

Lastly, you can export the Simulator recognized objects and object fields using the **Export Field Names** option. You can export the field names to a text for or to Excel.

Running Script Commands

The second feature of the Script Command Execution dialog is that you can run script commands manually. To run a script command, type the command into the display, and press the **Execute** button. Note that if the **Execute on ENTER key** option is checked, the command will also be processed when ENTER is pressed. Note that similar to script command syntax in the [Script](#) section of auxiliary files, script commands must be ended with a semi-colon (;) in the Script Command Execution dialog as well.

You can enter multiple script commands to be processed in sequence in this display. To do so, you must uncheck the option **Execute on ENTER key**. Then you can press enter after each script command to move to the next line and enter another command. Use the **Execute** button to process the sequence of script commands.

If you are running a sequence of script commands and wish to abort the run, use the **Abort** button.

If you wish to view the message log while script commands are processing, open the log using the **Show Log** button.

Quick Auxiliary Files Dialog

The Quick Auxiliary Files dialog can be accessed from the **Quick Aux Files** menu option of the [Script Command Execution](#) dialog. This dialog gives you a location for creating a list of auxiliary files to be processed en masse for the currently loaded case.

Using the **Define** option of the **Quick Aux Files** menu, the Quick Auxiliary Files dialog will open. Use this dialog to **Add** previously defined [auxiliary files](#) to the list of files to be processed. You can rearrange the order of the files by selecting a file and using the up and down arrows on the right to move the selected file within the list, or you can sort them alphabetically using the **Sort** button. To remove an auxiliary file from the list, use the **Delete** button.

Once the list of auxiliary files to process is complete, press the **Execute** button to process the list of auxiliary files.

Script Section

The SCRIPT section begins with a left curly brace and ends with a right curly brace.

SCRIPT ScriptName

```
{
script_statement_1
script_statement_2
.
.
.
script_statement_n
}
```

Scripts may optionally contain a ScriptName. This enables you to call a particular SCRIPT by using the [LoadScript action](#). After the optional name, the SCRIPT section begins with a left curly brace and ends with a right curly brace. Inside of this, script statements can be given.

In general, a script statement has the following format:

Keyword(arg1, arg2, ...);

- Statement starts with a keyword.
- The keyword is followed by an argument list which is encompassed in parentheses ().
- The arguments are separated by commas.
- If a single argument is a list of things, this list is encompassed by braces []. (eg. SetData)
- Statements end with a semicolon.
- Statements may take up several lines of the text file.
- You may put more than one statement on a single text line.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language. Versions of Simulator prior to 16.0 organized certain functionality with the submode feature. While existing scripts designed to work with submodes will function as before, moving between submodes is no longer necessary.

If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to **EDIT** mode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to **RUN** mode.

Script General Actions

The following actions are available to you regardless of the Mode:

RenameFile	("oldfilename",	"newfilename");		
CopyFile	("oldfilename",	"newfilename");		
DeleteFile	("filename");			
LoadAux	("filename",	CreateIfNotFound);		
LoadScript	("filename",	ScriptName,	CreateIfNotFound);	
LoadData	("filename",	DataName,	CreateIfNotFound);	
SelectAll	(objecttype,	filter);		
UnSelectAll	(objecttype,	filter);		
Delete	(objecttype,	filter);		
DeleteIncludingContents	(objecttype,	filter);		
SaveData("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList]);				
SaveDataWithExtra("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList], [Header_List], [Header_Value_List]);				
SaveDataUsingExportFormat("filename", filetype, FormatName, ModelToUse;				
SetData	(objecttype,	[fieldlist],	[valuelist],	filter);
CreateData	(objecttype,	[fieldlist],	[valuelist]);	
ChangeData	(objecttype,	[fieldlist],	[valuelist],	filter); (NOT AVAILABLE YET)
SaveObjectFields	("filename",	objecttype,	[fieldlist]);	
WriteTextToFile	("filename",	"text...");		
SetCurrentDirectory	("filedirectory",	CreateIfNotFound);		
SendtoExcel	(objecttype,	[fieldlist],	filter,	
	UseColumnHeaders,	"workbookname",	"worksheetname");	
StopAuxFile;				
ExitProgram;				
NewCase;				
OpenCase	("filename	// assumes		
	");	to open as		
		PWB		
OpenCase	("filename	openfiletyp		
	",	e);		
		PWB, GE,		
		PTI, or CF		
OpenCase	("filename	openfiletyp	[LoadTransactio	StarBus]);
	",	e,	ns,	

		PTI	YES NO DEFAULT [MSLine,	NEAR MAX 10000 VarLimDea d, Number	PostCTGAGC]); YES NO
OpenCase	("filename",	openfiletype, GE	MAINTAIN EQUIVALENCE		
AppendCase	("filename",	openfiletype); PWB, GE, PTI, or CF			
AppendCase	("filename",	openfiletype, PTI	[StarBus]); NEAR MAX <i>number</i> [MSLine,		
AppendCase	("filename",	openfiletype, GE	MAINTAIN EQUIVALENCE	VarLimDea d, Number	PostCTGAGC]); YES NO
SaveCase	("filename");		// assumes to save as PWB		
SaveCase	("filename",		savefiletype, PWB, PWB5, PWB6, PWB7,PWB8, PWB9, PWB10, PWB11, PWB12, PWB13 PTI23, PTI24, PTI25, PTI26, PTI27,PTI28, PTI29, PTI30 GE, GE14, GE15 CF		
EnterMode	[PostCTGAGC, YES or NO (mode); EDIT RUN		UseAreaZone]); YES or NO		
LogClear; LogSave	("filename",		AppendFile); YES or NO		
LogAdd	("string...");				
LogAddDateTime	("label",	IncludeDate, YES or NO	IncludeTime, YES or NO	IncludeMilliseconds); YES or NO	
CaseDescriptionClear; CaseDescriptionSet	("text...",		Append YES or NO		

SaveYbusInMatlabFormat	("filename",	IncludeVoltages); YES or NO				
SaveJacobian	("Jacfilename",	"JIDfilename",	filetype, M TXT	JacForm); R P		
SetParticipationFactors	(Method, MAXMWRAT RESERVE CONSTANT	ConstantValue, value	Object); [Area num] [Zone num] SYSTEM AREAZONE or DISPLAYFILTERS			
GenForceLDC_RCC	(filter);					
CalculateRXBGFromLengthConfigCondType	(filter);					
DirectionsAutoInsert	(Source, AREA ZONE INJECTIONGROUP	Sink, AREA ZONE INJECTIONGROUP SLACK	DeleteExisting, YES or NO	UseDisplayFilters, YES or NO		Save
DeterminePathDistance	([start],	BranchDistMeas,	BranchFilter,	BusField);		
DetermineShortestPath	([start]	[end],	BranchDistMeasure,	BranchFilter,	FileName);	
ChangeSystemMVA Base	(NewBase);					
AutoInsertTieLineTransactions;						
InjectionGroupsAutoInsert;						
DetermineBranchesThatCreateIslands	(BranchFilter, ALL Selected Closed "FilterName" (filter);	StoreBuses, YES or NO	"filename",	SetSelectedOnLines); YES or NO		
SetGenPMaxFromReactiveCapabilityCurve						
SetScheduledVoltageForABus	([bus identifier],	voltage);				
Scale	(scaletype,	basedon,	[parameters],	ScaleMarker);		

LOAD GEN	MW FACTOR	[P,Q] [P] (means constant pf)	BUS AREA
INJECTIONGROUP BUSSHUNT		[P, +Q, -Q]	ZONE OWNER

RenameFile("oldfilename", "newfilename");

Use this action to rename a file from within a script.

"oldfilename": The present file name.

"newfilename": The new file name desired.

CopyFile("oldfilename", "newfilename");

Use this action to copy a file from within a script.

"oldfilename": The present file name.

"newfilename": The new file name desired.

DeleteFile("filename");

Use this action to delete a file from within a script.

"filename": The file name to delete.

LoadAux("filename", CreateIfNotFound);

Use this action to load another auxiliary file from within a script.

"filename": The filename of the auxiliary file being loaded.

CreateIfNotFound: Set to YES or NO. YES means that objects that cannot be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

LoadScript("filename", ScriptName, CreateIfNotFound);

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

"filename": The filename of the auxiliary file being loaded.

ScriptName: The specific ScriptName from the auxiliary file which should be loaded.

CreateIfNotFound: Set to YES or NO. YES means that objects which can not be found will be created while reading in SCRIPT sections from filename. If this parameter is not specified, then NO is assumed.

LoadData("filename", DataName, CreateIfNotFound);

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

"filename": The filename of the auxiliary file being loaded.

DataName: The specific ScriptName from the auxiliary file which should be loaded.

CreateIfNotFound: Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

SelectAll(objecttype, filter);

Use this to set the selected property of objects of a particular type to true. A filter may optionally be specified to only set this property for objects which meet a filter.

objecttype: The objecttype being selected.

filter: There are three options for the filter:

SelectAll(objecttype);: No filter specified means to select all objects of this type.

SelectAll(objecttype, "filtername");: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SelectAll(objecttype, AREAZONE);: AREAZONE means select those that meet the area/zone filters.

UnSelectAll(objecttype, filter);

Same as SelectAll, but this action sets the Selected field to false.

Delete(objecttype, filter);

Use this to delete objects of a particular type. A filter may optionally be specified to only delete objects that meet a filter.

objecttype: The objecttype being selected.

filter: There are four options for the filter:

Delete(objecttype);: No filter specified means to delete all objects of this type.

Delete(objecttype, "filtername");: "filtername" means delete those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Delete(objecttype, AREAZONE);: AREAZONE means delete those that meet the area/zone filters.

Delete(objecttype, SELECTED);: SELECTED means delete those objects whose Selected field is set to *YES*.

DeleteIncludingContents(objecttype, filter);

Use this to delete objects of a particular type and other objects that these contain. Currently, only multi-section lines (objecttype = MultiSectionLine) can be used with this command. The branches and dummy buses that belong to multi-section lines will also be deleted along with the multi-section lines. A filter may optionally be specified to only delete objects that meet a filter.

objecttype: The objecttype being selected (presently we only support objecttype = MultiSectionLine).

filter: There are four options for the filter:

DeleteIncludingContents(objecttype);: No filter specified means to delete all objects of this type.

DeleteIncludingContents(objecttype, "filtername");: "filtername" means delete those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

DeleteIncludingContents(objecttype, AREAZONE);: AREAZONE means delete those that meet the area/zone filters.

DeleteIncludingContents(objecttype, SELECTED);: SELECTED means delete those objects whose *selected* field is set to YES.

SaveData("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList]);

Use this action to save data in a custom defined format. A filter may optionally be specified to save only objects which meet a filter, and a sort order list may also optionally be specified that will allow saving the objects in a particular order.

"filename": The file to save the data to

filetype:

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

CSVColHeader: save as a normal CSV file without the AUX file syntax, and with the first row showing column headers you would see in a case information display.

objecttype: The objecttype being saved.

[fieldlist]: A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, *variablename:location:digits:rod*.

[subdatalist]: A list of the subdata objecttypes to save with each object record.

filter: There are four options for the filter:

SaveData(...);: No filter specified means to save all objects of this type.

SaveData(..., "filtername");: "filtername" means save those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SaveData(..., AREAZONE);: AREAZONE means save those that meet the area/zone filters.

SaveData(..., SELECTED);: SELECTED means save those objects whose *selected* field is set to YES.

[SortFieldList]: This allows the specification of a sort order in which the data will be saved. The format is:

[variablename1+:0, variablename2-:1]

where

variablename is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, *variablename* can be in the format **fieldname:location**.

+ or **-** for the second parameter indicates sort ascending for **+** and sort descending for **-**. This parameter must be specified.

0 or **1** for the third parameter indicates: **0** - case insensitive and do not use absolute value, **1** - case sensitive or use absolute value. This parameter is optional.

SaveDataWithExtra("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList], [Header_List], [Header_Value_List]);

Use this action to save data in a custom defined format. User-specified fields and field values can also be specified in the output. Optional parameters are filter, SortFieldList, Header_List, and Header_Value_List.

"filename": The file to save the data to

filetype:

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

CSVNOHEADER: save as a normal CSV text file, without the AUX file formatting. The object name and field names are NOT included. This option is useful when appending data of the same object type and field list into a common file.

CSVColHeader: save as a normal CSV file without the AUX file syntax, and with the first row showing column headers you would see in a case information display.

Data cannot be saved using the AUX filetype with this command.

objecttype: The objecttype being saved.

[fieldlist]: A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, **variablename:location:digits:rod**.

[subdatalist]: A list of the subdata objecttypes to save with each object record.

filter: There are four options for the filter:

SaveData(...);: No filter specified means to save all objects of this type.

SaveData(..., "filtername");: "filtername" means save those that meet the filter.

SaveData(..., AREAZONE);: AREAZONE means save those that meet the area/zone filters.

SaveData(..., SELECTED);: SELECTED means save those objects whose *selected* field is set to YES.

[SortFieldList]: This allows the specification of a sort order in which the data will be saved. The format is:

[variablename1:+:0, variablename2:-:1]

where

variablename is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, **variablename** can be in the format **fieldname:location**.

+ or **-** for the second parameter indicates sort ascending for **+** and sort descending for **-**. This parameter must be specified.

0 or **1** for the third parameter indicates: **0** - case insensitive and do not use absolute value, **1** - case sensitive or use absolute value. This parameter is optional. If it is not specified, the default is 0.

[Header_List]: This allows the specification of user-defined fields that will appear in the output. Headers should be specified as a list of comma delimited strings. A string should be enclosed in double quotes if the string contains a comma. Header strings cannot be blank.

[Header_Value_List]: Allows the specification of the values that should be assigned to the user-defined fields specified by Header_List. Specifying the values is optional. If specified, there must be as many values specified as there are headers. If not specified, all values are blank. Each object will use the same specified value for the specified field. To use different values for

different objects and save these in the same file, make use of the CSVNOHEADER file format and filtering.

SaveDataUsingExportFormat("filename", filetype, FormatName, ModelToUse);

Use this action to save data in a user-defined format that has previously been defined.

"filename": The file to save the data to

filetype:

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

FormatName: The name of the Object Export Format Description to use.

ModelToUse: Optional parameter that indicates the model to use.

FULL: Full-topology model. This is the default if the parameter is omitted.

CONSOLIDATED: Consolidated planning-type model. This option will only work with the Topology Processing add-on.

SetData(objecttype, [fieldlist], [valuelist], filter);

Use this action to set fields for particular objects. If a filter is specified, then it will set the respective fields for all objects which meet this filter. Otherwise, if no filter is specified, then the keyfields must be included in the field list so that the object can be found.

objecttype: The objecttype being set.

[fieldlist]: A list of fields that you want to save.

[valuelist]: A list of values to set the respective fields to.

filter: There are four options for the filter:

SetData(...);: No filter specified: set data only for the object described by the [fieldlist] and [valuelist] parameters.

SetData(..., "filtername");: "filtername": set data for all objects that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SetData(..., AREAZONE);: AREAZONE: set data for all objects that meet the area/zone filters.

SetData(..., SELECTED);: SELECTED: set data for all objects whose *selected* field is set to YES.

SetData(..., ALL);: ALL: set data for all objects of the specified object type.

CreateData(objecttype, [fieldlist], [valuelist]);

Use this action to create particular objects. Note that the key fields for the objecttype must be specified.

objecttype: The objecttype being set.

[fieldlist]: A list of fields that you want to save.

[valuelist]: A list of values to set the respective fields to.

SaveObjectFields("filename", objecttype, [fieldlist]);

Use this action to save a list of fields available for the specified objecttype to a CSV file. Format of the file is *variablename, field, col header, description*.

"filename": The file path and name to save.

objecttype: The type of object for which fields should be saved.

[fieldlist]: List of fields for which information will be saved.

WriteTextToFile("filename", "text...");

Use this action to write text to a file. If the specified file already exists, the text will be appended to the file. Otherwise, it creates the file and writes the text to the file.

"filename": The file path and name to save.

"text...": The text to be written to the file.

SetCurrentDirectory("filedirectory", CreateIfNotFound);

Use this action to set the current working directory.

"filedirectory": The path of the work directory.

CreateIfNotFound: Set to YES or NO. YES means that if the directory path can not be found, the directory will be created. If this parameter is not specified, then NO is assumed.

SendtoExcel(objecttype, [fieldlist], filter, UseColumnHeaders, "workbookname", "worksheetname");

Use this action to mimic the behavior of Send to Excel option within a case information display.

objecttype : The type of object for which fields should be saved.

[fieldlist] : List of fields for which information will be saved.

filter : There are four options for the filter:

SendToExcel(...) : No filter specified means to save all objects of this type.

SendToExcel (... , "filtername") : "filtername" means save those meeting the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SendToExcel (... , AREAZONE) : AREAZONE means save those that meet the area/zone/owner filters.

SendToExcel (... , SELECTED) : SELECTED means save those that are selected.

UseColumnHeaders : Set to YES or NO. YES signifies that the first row shows the Column Header, NO signifies that variable names are used.

"workbookname" : Optional parameter to specify the path and name to save. If blank, a new workbook will be created, if a value is specified it will overwrite any existing file of that name.

"worksheetname" : Optional parameter to specify the worksheet name to save. If blank, a new worksheet will be created, if a value is specified it will overwrite the data in any existing worksheet of that name.

StopAuxFile;

Use this action to treat the remainder of the file after the command as a big comment. This includes any script commands inside the present SCRIPT block, as well as all remaining SCRIPT or DATA blocks.

ExitProgram;

Immediately exits the program with no prompts.

NewCase;

This action clear out the existing case and open a new case from scratch.

OpenCase("filename", OpenFileType);

OpenCase("filename", OpenFileType,[LoadTransactions, StarBus]);

OpenCase("filename", OpenFileType,[MSLine, VarLimDead, PostCTGAGC]);

This action will open a case stored in "filename" of the type OpenFileType;

"filename": The file to be opened.

OpenFileType: An optional parameter indicating the format of the file being opened. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PTI, GE, CF, AUX.

[LoadTransactions, StarBus]: An optional set of parameters used ONLY when reading in a PTI raw file.

LoadTransactions:

YES: Creates MW transactions, replicating the option (AREA INT CODE = 1 FOR LINES ONLY)

NO: Do not create, replicating the option (AREA INT CODE = 2 FOR LINES ONLY)

DEFAULT: Let the software estimate which is best

StarBus:

NEAR: Use numbers near the primary bus number for star bus assignment

MAX: Start numbering star buses above the maximum bus number in the case

Number: Start numbering at the number specified

[MSLine, VarLimDead, PostCTGAGC]: An optional set of parameters used ONLY when reading in a GE epc file.

MSLine:

MAINTAIN: keeps the multi-section lines as defined in the epc file

EQUIVALENCE: removes the multi-section lines by creating a single equivalent line combining the sections of each line

VarLimDead:

Number: The assumed generator VAR limit deadband

PostCTGAGC:

YES: prevents generators from responding to system changes during a contingency outage solution

NO: all generators to respond to system changes during a contingency outage solution

AppendCase("filename", OpenFileType);

AppendCase("filename", OpenFileType, [StarBus]);

AppendCase("filename", OpenFileType, [MSLine, VarLimDead, PostCTGAGC]);

This action will open and [append a case](#) stored in "filename" of the type OpenFileType to the load flow case currently loaded in Simulator.

"filename": The file to be opened and appended.

OpenFileType: An optional parameter indicating the format of the file being opened. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PTI, GE, CF, AUX.

[StarBus]: An optional set of parameters used ONLY when reading in a PTI raw file.

StarBus:

NEAR: Use numbers near the primary bus number for star bus assignment

MAX: Start numbering star buses above the maximum bus number in the case

Number: Start numbering at the number specified

[MSLine, VarLimDead, PostCTGAGC]: An optional set of parameters used ONLY when reading in a GE epc file.

MSLine:

MAINTAIN: keeps the multi-section lines as defined in the epc file

EQUIVALENCE: removes the multi-section lines by creating a single equivalent line combining the sections of each line

VarLimDead:

Number: The assumed generator VAR limit deadband

PostCTGAGC:

YES: prevents generators from responding to system changes during a contingency outage solution

NO: all generators to respond to system changes during a contingency outage solution

SaveCase("filename", SaveFileType, [PostCTGAGC, UseAreaZone]);

This action will save the case to "filename" in the format SaveFileType.

"filename": The file name to save the information to.

SaveFileType: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PWB5, PWB6, PWB7, PWB8, PWB9, PWB10, PWB11, PWB12, PWB13

PTI (means PTI30), PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30

GE (means GE15), GE14, GE15

CF

AUX

PostCTGAGC : An optional parameter, only valid for GE EPC format. YES sets the base load flag in the EPC file based on the Post-Contingency Prevent AGC Response setting. If preventing post-contingency AGC, the base load flag is set to 1. If not preventing post-contingency AGC or this option is set to NO, the base load flag is set to 0.

UseAreaZone : An optional parameter, only valid for GE EPC format. YES limits the entries in the EPC file based on the area/zone/owner filter (NO by default)

EnterMode(mode);

This action will tell Simulator what mode to enter before performing the next script command.

Mode: The mode to enter, RUN or EDIT.

LogClear;

Use this action to clear the Message Log.

LogSave("filename", AppendFile);

This action saves the contents of the Message Log to "filename".

"filename": The file name to save the information to.

AppendFile: Set to YES or NO. YES means that the contents of the log will be appended to "filename". NO means that "filename" will be overwritten.

LogAdd("string...");

Use this action to add a personal message to the MessageLog.

"string...": The string that will appear as a message in the log.

LogAddDateTime("label", IncludeDate, IncludeTime, IncludeMilliseconds);

Use this action to add a personal message to the MessageLog, including the posting date/time.

"label": A string that will appear at the start of the line containing the date/time.

IncludeDate: Set to YES or NO. YES means that the date will be added after the string specified. If this parameter is not specified, then YES is assumed.

IncludeTime: Set to YES or NO. YES means that the time will be added after the date. If there's no date, then it will be added after the string specified. If this parameter is not specified, then YES is assumed.

IncludeMilliseconds: Set to YES or NO. YES means that the time including milliseconds will be added after the date or string specified. This is valid only if the IncludeTime is set to YES. If this parameter is not specified, then YES is assumed.

CaseDescriptionClear;

Use this action to clear the case description of the presently open case.

CaseDescriptionSet("text...", Append);

Use this action to set or append text to the case description

"text...": Specify the text to set/append to the case description

Append: YES will append the text to the existing case description. NO will replace the case description

SaveYbusInMatlabFormat("filename", IncludeVoltages);

Use this action to store the power system bus admittance matrix (Ybus) in a text format that can be easily read into other programs such as MATLAB®.

"filename": The file name to save the information to. The Ybus data is stored using the MATLAB sparse matrix format in the matrix Ybus.

IncludeVoltages: Set to YES or NO. YES means that the bus voltages are also stored but in the vector V.

SaveJacobian("filename", IncludeVoltages);

Use this action to store the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®.

"Jacfilename": The file name to save the Jacobian data to. The Jacobian is stored using the MATLAB sparse matrix format in the matrix Jac.

"JIDfilename": The file name to save the text identifier information to. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

filetype: A parameter saying the format of the file to be saved. It may be one of the following strings:

M: MATLAB .M Format.

TXT: ASCII format.

JacForm: Specifies the form of the Jacobian. It may be one of the following strings:

R: Rectangular form

P: Polar form.

SetParticipationFactors(Method, ConstantValue, Object);

Use this action to define the participation factors for multiple generators.

Method: The formula used to calculate the participation factors for each generator. It may be one of the following strings:

MAXMW RAT: The participation factor for each generator is set to the generator's maximum MW capability.

RESERVE: The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute (Max MW rating - Present MW).

CONSTANT: The participation factor for each generator is set to the same specified value.

ConstantValue: The value used if CONSTANT method is specified. If CONSTANT method is not specified, enter 0 (zero).

Object: The set of generators to which the participation factors will be assigned. It may be one of the following settings:

[Area num], [Area "name"] [Area "label"]: For all the generators in area

[Zone num], [Zone "name"] [Zone "label"]: For all the generators in zone

SYSTEM: For all the generators in the system regardless of their area or zone affiliation.

AREAZONE or DISPLAYFILTERS: For just the generators whose area/zone/owner filter criteria evaluates to true.

GenForceLDC_RCC(filter);

Use this action to convert the voltage setpoint for Line Drop and Reactive Current Compensation (LDC/RCC) for multiple generators.

filter: There are four options for the filter:

GenForceLDC_RCC;; No filter specified means to convert voltage setpoint for LDCC/RCC on all the generators of the system.

GenForceLDC_RCC ("filtername");; "filtername" means to convert voltage setpoint for LDCC/RCC on those generators that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

GenForceLDC_RCC (AREAZONE);; AREAZONE means to convert voltage setpoint for LDCC/RCC on those generators that meet the area/zone filters.

GenForceLDC_RCC (SELECTED);; SELECTED means to convert voltage setpoint for LDCC/RCC on those generators whose *selected* field is set to YES.

CalculateRXBGFromLengthConfigCondType(filter);

Use this action to go through branches in the power system and automatically recalculate the per unit R, X, G, and B values using the TransLineCalc tool. The branches Conductor Type, Tower Configuration, and Line Length will be passed to the TransLineCalc tool and new R, X, G and B values will be calculated. This is only available if you have installed the TransLineCalc tool.

filter: There are four options for the filter which operate identically to the option for the GenForceLDC_RCC action above.

CalculateRXBGFromLengthConfigCondType;; No filter specified means to perform calculation on all branches.

CalculateRXBGFromLengthConfigCondType ("filtername");; "filtername" means to perform calculation on those branches that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

CalculateRXBGFromLengthConfigCondType;; AREAZONE means to perform calculation on those branches that meet the area/zone filters.

CalculateRXBGFromLengthConfigCondType; SELECTED means to perform calculation on those branches whose *selected* field is set to YES.

DirectionsAutoInsert(Source, Sink, DeleteExisting, UseDisplayFilters, Start, Increment);

Use this action to automatically insert multiple directions for PTDF studies.

Source, Sink: The type of object used as the source/sink of the directions. Currently, only the following types of directions can be automatically defined:

AREA SLACK: Directions from areas to slack bus of the system.

ZONE SLACK: Directions from zones to slack bus of the system.

INJECTIONGROUP SLACK: Directions from injection groups to slack bus of the system.

AREA AREA: Directions from areas to other areas of the system.

ZONE ZONE: Directions from zones to other zones of the system.

INJECTIONGROUP INJECTIONGROUP: Directions from injection groups to other injection groups of the system.

DeleteExisting: Set to YES or NO. YES means that previously defined directions will be deleted before the new directions are automatically inserted. NO means automatically inserted directions will be added to the list of previously defined directions. If this parameter is not specified, then YES is assumed.

UseDisplayFilters: Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting directions. NO specifies all Areas and Zones will be used to automatically insert directions regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

Start: Integer value that specifies the first number used when automatically numbering the automatically inserted directions. If this parameter is not specified, then 1 is assumed.

Increment: Integer value that will be used as increment in numbering the subsequent automatically inserted directions. If this parameter is not specified, then 1 is assumed.

DeterminePathDistance([start], BranchDistMeas, BranchFilter, BusField);

Use this action to calculate a distance measure at each bus in the entire model. The distance measure will represent how far each bus is from the starting group specified. The distance measure can be related to impedance, geographical distance, or simply the number of nodes.

[start] : is an the starting place which is enclosed in brackets. The starting place may be either an Area, Zone, SuperArea, Substation, Injection Group, or Bus.

BranchDistMeas : is either X, Z, Length, Nodes, or a variable name for a branch.

X : means use the series reactance,

Z : means use $\sqrt{X^2 + R^2}$,

Length : means us the Length field, and

Nodes : means treat each branch as a length of one.

"variablename" : Otherwise use any Branch object variable name.

BranchFilter : is either All, Selected, Closed or the name of a branch Advanced Filter. This parameter is used to specify which branch can be traversed at all.

All : means all branches can be traversed

Selected : means only branch whose Selected field is YES can be traversed

Closed : means only branches that are CLOSED can be traversed.

"Filtername" : means only branches who meet the Advanced Filter named can be traversed. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

BusField : is the variablename of a Bus field. This field is populated with the minimum distance from the Start Place to that bus. All buses in the start group will have a distance measure of zero. Buses which can not be reached from the start group will have a distance measure of -1.

DetermineShortestPath([start], [end], BranchDistanceMeasure, BranchFilter, Filename);

Use this action to calculate the shortest path between a starting group and an ending group. The results will be written to a textfile specified by filename. In the text file, the first bus listed will be in the end grouping and the last bus listed will be the start grouping. The result text file will have a line for each bus passed. Each line will contain three entries delimited by a space: "Number DistanceMeasure Name".

[start] : same as the starting place for the DeterminePathDistance script command
[end] : same as the starting place for the DeterminePathDistance script command
BranchDistanceMeasure : same as for DeterminePathDistance script command
BranchFilter : same as for DeterminePathDistance script command
Filename : is a filename (may need to be enclosed in quotes) to which the results will be written.

ChangeSystemMVABase(NewBase);

Use this action to change the system MVA base to the specified value and update all internal data structures to store values on the new base.

NewBase: New power system base in MVA.

AutoInsertTieLineTransactions;

Use this action to delete all existing MW transactions and set the unspecified MW interchange for each area to zero. It then automatically creates a MW transaction between each pair of connected areas with a MW transaction exactly equal to the sum of the tie-line flows

InjectionGroupsAutoInsert;

Use this action to automatically insert injection groups according to the options specified in the "IG_AutoInsert_Options" object. The settings available with this object represent what is seen on the Auto Insert Injection Groups Dialog.

DetermineBranchesThatCreateIslands(BranchFilter, StoreBuses, "filename", SetSelectedOnLines);

Use this action to determine the branches whose outage results in island formation. Note that setting the Selected field will overwrite the Selected fields.

BranchFilter : is either All, Selected, Closed or the name of a branch Advanced Filter. This parameter is used to specify which branch can be traversed at all.

All : means all branches can be traversed

Selected : means only branches whose Selected field = YES can be traversed

Closed : means only branches that are CLOSED can be traversed.

"FilterName" : means only branches who meet the advanced filter named can be traversed. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

StoreBuses : YES to store the buses in the island to the output file

"filename" : file to which the results will be written.

SetSelectedOnLines : YES to set the SELECTED field to YES for branches that create islands

SetGenPMaxFromReactiveCapabilityCurve(filter);

Use this action to change the Maximum MW output of generators which use a capability curve, equal to the second-to-last MW point in the capability curve if the last Max Mvar point on the capability curve is smaller than 0.001 Mvar. If the present MW output is higher than this new Max MW value, then Max MW is set to the present MW output..

filter : optional parameter which is either Selected, Closed or the name of a branch Advanced Filter. This parameter is used to specify which generators are processed. If blank, all generators are processed.

Selected : means only generators whose Selected field = YES can be processed

AREAZONE : means process those generators that meet the area/zone/owner filters.

"FilterName" : means only generators who meet the advanced filter named can be processed. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SetScheduledVoltageForABus([bus identifier], voltage);

Use this action to set the stored scheduled voltage, vsched, for a bus according to how this is defined in the EPC format. This value is not used by Simulator but is stored for purposes of writing out to an EPC file. The setpoint voltages for generators and switched shunts regulating the specified bus are also set to the new voltage. The regulation range for switched shunts is modified for the new setpoint voltage according to how this is defined in the EPC format: $vband = (VHigh - VLow)/2$ with newVHigh = voltage+vband and new VLow = voltage-vband.

[bus identifier] : specifies bus

voltage : the new voltage

Scale(scaletype, basedon, [parameters], scalemarker);

Use this action to scale the load and generation in the system.

scaletype: The objecttype begin scaled. Must be either LOAD, GEN, INJECTIONGROUP, or BUSSHUNT.

basedon: MW: parameters are given in MW, MVAR units.

FACTOR: parameters a factor to multiple the present values by.

[parameters]: These parameters have different meanings depending on ScaleType.

LOAD: [MW, MVAR] or [MW]. If you want to scale load using constant power factor, then do not specifying a MVAR value.

GEN: [MW].

INJECTIONGROUP: [MW, MVAR] or [MW] . If you want to scale load using constant power factor, then do not specifying a MVAR value.

BUSSHUNT: [GMW, BCAPMVAR, BREAMVAR]. The first values scales G shunt values, the second value scales positive (capacitive) B shunt values, and the third value scales negative (reactive) B shunt values.

scalemarker: This value specifies whether to look at an element's bus, area, or zone to determine whether it should be scaled.

BUS: Means that elements will be scaled according to the BusScale property of the element's terminal bus.

AREA: Means that elements will be scaled according to the BGScale property of the element's Area. Note that it is possible for the area of a load, generator, or switched shunt to be different than the terminal bus's area.

ZONE: Means that elements will be scaled according to the BGScale property of the element's Zone. Note that it is possible for the zone of a load, generator, or switched shunt to be different than the terminal bus's zone.

OWNER: Means that the elements will be scaled according to the BGScale property of the element's Owner.

Script Online Actions

OpenOneLine	("filename	"view",	FullScreen,	Show	
	",			Full);	
CloseOneLine	("Online				
	Name");				
SaveOnline	("filename	"Online	SaveFileType);		
	",	Name",			
ExportOnlineAs	("filename	"Online	"ShapeFileExportDescri	UseLo	PointLoc
ShapeFile	",	Name",	ptionName",	nLat,	ation);
LoadAXD	("filename	"Online	CreateIfNotFound);		
	",	Name",			

OpenOnline("filename", "view", FullScreen);

Use this action to open a online diagram. This can only be used in SimAuto and cannot be used to actually view a online. This action is most useful when wanting to save a online diagram in a different file format.

"filename": The file name of the online diagram to open.

"view": The view name that should be opened. Pass an empty string to denote no specific view.

FullScreen: Set to YES or NO. YES means that the online diagram will be opened in full screen mode. If this parameter is not specified, then NO is assumed.

ShowFull: Optional parameter. Set to YES to open the online and apply the Show Full option. Set to NO to open the online and leave the online as is. Default is NO if not specified.

CloseOnline("OnlineName");

Use this action to close an open online diagram without saving it. If the name is omitted, the last focused online diagram will be closed.

"OnlineName": The name of the online diagram to close.

SaveOnline("filename", "OnlineName", SaveFileType);

Use this action to save an open online diagram to a file.

"filename": The path and file name of the file to save.

"OnlineName": Name of the open online to save.

SaveFileType: Type of file to save. Valid options are PWB, PWB5, PWB6, PWB7, PWB8, PWB9, PWB10, PWB11, PWB12, PWB13. If omitted, PWB, which is the most recent version, will be assumed.

ExportOnlineAsShapeFile("filename", "OnlineName", "ShapeFileExportDescriptionName", UseLonLat, PointLocation);

Use this action to save an open online diagram to a shapefile.

"filename": The file name of the shapefile to save.

"OnlineName": The name of the online diagram to save to a shapefile. The online diagram must be open. Use the OpenOnline script command if necessary to open the appropriate online.

"ShapeFileExportDescriptionName": Name of the ShapeFile Export Description to use when saving the shapefile.

UseLonLat: Set to YES or NO. YES means that the coordinates of objects on the online diagram will be saved using longitude,latitude. This will only be true if a valid map projection is in use with the online diagram. Otherwise, the coordinates will be saved in x,y. If this parameter is set to NO, the coordinates will be saved in x,y. If this parameter is not specified, YES is assumed.

PointLocation: Determines where points are specified -- object centers, or the upper left corner. Specify "center" to define points as the shape centers, or "ul" to define them as the upper left corner of the shapes. If not specified, upper left is assumed.

LoadAXD("filename", "OnlineName", CreateIfNotFound);

Use this action to apply a display auxiliary file to an open online diagram. This action is only allowed when using SimAuto.

"filename": The file name of the display auxiliary file to load.

"OnlineName": The name of the online diagram to which to apply the display auxiliary file. Use the OpenOnline script command if necessary to open the appropriate online. If the specified online is not open, a new one will be created with the given name.

CreateIfNotFound: Set to YES or NO. YES means that objects which cannot be found will be created while reading in DATA sections from filename.

Script Edit Mode Actions

The following case related script commands are available in Edit mode:

Equivalence;
DeleteExternalSystem;

SaveExternalSystem ("filename", Savefiletype, withties);

Move ([elementA], [destination parameter], percentage);
[GEN numA idA]
[LOAD numA idA] [numB idB]
[BRANCH numA1 numA2 cktA] [numB1 numB2 cktB]
[SHUNT numA idA] [numB idB]

Combine ([elementA], [elementB]);
[GEN numA idA] [GEN numB idB]
[LOAD numA idA] [LOAD numB idB] // NOT AVAILABLE YET
[BRANCH numA1 numA2 cktA] [BRANCH numB1 numB2 cktB] // NOT AVAILABLE YET

SplitBus ([element], NewBusNumber, InsertBusTieLine, LineOpen);
[BUS num] num YES or NO YES or NO

MergeBuses ([element], filter);
[BUS num]

MergeLineTerminals (filter);

TapTransmissionLine ([element], PosAlongThe Line, NewBusNumber, ShuntModel, TreatAsMSLine);
[BRANCH numA1 numA2 cktA]; Value in % number, num LINESHUNTS YES or NO
CAPACITANCE

```

RenumberMSLineDummyBuses    ("filename");

Renumber3WXFormerStarBuses    ("filename");

InterfacesAutoInsert          (Type      DeleteExisting    UseDisplayFilters    Prefix,    Limits);
                              'AREA      'YES or NO        'YES or NO        "string    AUTO
                              'ZONE                                     "            ZEROS
                                                                [value1
                                                                ' ...
                                                                value
                                                                8]

```

Equivalence

This action will equivalence a power system. All options regarding equivalencing are handled by the Equiv_Options objecttype. Use the SetData() action, or a DATA section to set these options prior to using the Equivalence() action. Also, remember that the property BusEquiv must be set true for each bus that you want to equivalence.

DeleteExternalSystem

This action will delete part of the power system. It will delete those buses whose property BusEquiv must is set true.

SaveExternalSystem("filename", SaveFileType, WithTies)

This action will save part of the power system to a "filename". It will save only those buses whose property BusEquiv must is set true.

"filename": The file name to save the information to.

SaveFileType: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. May be one of the following strings:

PWB, PWB5, PWB6, PWB7. PWB8, PWB9, PWB10, PWB11, PWB12, PWB13, PWB14
PTI (means PTI30) PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30, PTI31, PTI32
GE (means GE15), GE14, GE15, GE16, GE17
CF
AUX

WithTies: An optional parameter. The user must specify the file type explicitly in order to use the WithTies parameter. Allows saving transmission lines that tie a bus marked with BusEquiv as false and one marked true. This must be a string which starts with the letter Y, otherwise NO will be assumed.

Move([elementA], [destination parameters], percentage);

Use this action to move a generator, load, switched shunt, or transmission line.

[elementA]: the object that should be moved. Must be one of the following formats.

[GEN busnum id], [GEN "name_nomkv" id], [GEN "buslabel" id], [GEN "label"]
 [LOAD busnum id], [LOAD "name_nomkv" id], [LOAD "buslabel" id], [LOAD "label"]
 [BRANCH busnum1 busnum2 ckt], [BRANCH "name_kv1" "name_kv2" ckt],
 [BRANCH "buslabel1" "buslabel2" ckt], [BRANCH "label"]
 [SHUNT busnum id], [SHUNT "name_nomkv" id], [SHUNT "buslabel" id], [SHUNT "label"]

[destination parameters]: These parameters have different meanings depending on object type of the element. Bus numbers must be used here.

GEN: [busnum id]

LOAD: [busnum id]

BRANCH: [busnum1 busnum2 id]

SHUNT: [busnum id]

percentage: The percentage of the device to move to the new destination. This is only valid for generators and loads, and allows partial generation or load to be moved to the new location.

Combine([elementA], [elementB]);

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS.

Use this action to combine two generators, two loads, or two transmission line. Note that elementA and elementB must be of the same object type. You can not combine a BRANCH and a LOAD.

[elementA]: The object that should be moved. Must be one of the following formats.

[GEN busnum id]

[LOAD busnum id]

[BRANCH busnum1 busnum2 ckt]

[elementB]: The object that element A should be combined with. Must the same format as for elementA.

SplitBus([element], NewBusNumber, InsertBusTieLine, LineOpen);

Use this action to transform one bus into two connected buses.

[element]: The bus that should be split. Must be of the following format:

[BUS num]

[BUS "name_nomkv"]

[BUS "buslabel"]

NewBusNumber: The number of the new bus resulting from the split. Must be unique, that is, it cannot be a number that identifies another number in the case.

InsertBusTieLine Set to YES or NO. YES means to insert a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of 0.0000 + j0.0001 ohms. NO means that no bus tie will be inserted. If this parameter is not specified, then YES is assumed.

LineOpen Set to YES or NO. YES means that the bus tie should be inserted as an open branch. NO means that the bus tie will be inserted as a closed branch. This options is valid only if InsertBusTieLine is set to YES. If this parameter is not specified, then NO is assumed.

MergeBuses([element], filter);

Use this action to merge a set of buses into a single bus.

[element]: The number of the new bus resulting from the merging. It should be a number of a bus that is part of the set of buses to be merged, or it should be unique, meaning that it cannot be a number that identifies another number in the case. Must be of the following format:

[BUS num]

filter: There are four options for the filter:

MergeBuses(...);: No filter specified means to merge all the buses of the system.

MergeBuses(..., "filtername");: "filtername" means to merge those buses that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

MergeBuses(..., AREAZONE);: AREAZONE means to merge those buses that meet the area/zone filters.

MergeBuses(..., SELECTED);: SELECTED means to merge those buses whose *selected* field is set to YES.

MergeLineTerminals(filter);

Use this action to remove a line or lines from the model by combining the terminal buses at both ends of a line into a single bus.

filter: There are two options for the filter:

MergeLineTerminals("filtername");: "filtername" means to individually merge the terminal buses of each transmission line that meets the advanced filter.

MergeLineTerminals(SELECTED);: SELECTED means to individually merge the terminals of each transmission line whose *selected* field is set to YES.

TapTransmissionLine([element], PosAlongTheLine, NewBusNumber, ShuntModel, TreatAsMSLine);

Use this action to insert a bus at some location along an existing transmission line.

[element]: The transmission line to tap. Must be of the following format:

[BRANCH busnum1 busnum2 ckt]

[BRANCH "name_kv1" "name_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

PosAlongTheLine: The value that indicates the point where the new bus is to be inserted relative to the location of the nearest of the two buses in terms of a percentage of the total line length. This value controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplies by the impedance of the original line.

NewBusNumber: The number of the new bus resulting from the tapping. Must be unique, that is, it cannot be a number that identifies another number in the case.

ShuntModel: The way the original charging capacitance of the line is reassigned. The choices are:

LINESHUNTS: The original charging capacitance is reassigned as line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new branch elements will be set to 0.

CAPACITANCE: The original charging capacitance of the line is converted approximately using percentage entered as PosAlongTheLine. In other words, the original capacitance will be scaled and assigned to the new segments as charging capacitance.

The default ShuntModel is LINESHUNTS.

TreatAsMSLine: Set to YES or NO. YES means that a new multi-section line will be created that is comprised of the two new lines. NO means that the two new lines will be controlled independently of each other.

RenumberMSLineDummyBuses("filename");

Use this action to change the dummy bus numbers and names for one or more multi-section lines to new numbers and names of your choosing. The new dummy bus numbers and names are loaded from a file. The format of the data in the file is described in the help on the [Multi-Section Lines Display](#).

"filename": The file name of the file containing the specification of the new dummy bus numbers and names.

Renumber3WXFormerStarBuses("filename");

Use this action to change the star bus number and name for one or more three winding transformer star buses to new numbers and names of your choosing. The new star bus numbers and names are loaded from a file. The format of the data in the file is described in the help on the [three winding transformer display](#).

"filename": The file name of the file containing the specification of the new star bus numbers and names.

InterfacesAutoInsert(Type, DeleteExisting, UseDisplayFilters, Prefix, Limits);

Use this action to automatically insert a group of interfaces. The new of the new interfaces defaults to "Area1-Area2" or "Zone1-Zone2" with an optional **Prefix**.

Type: The type of interfaces to insert. Choices are:

AREA: Area-to-area interfaces join adjacent areas (those that share at least one tie line).

ZONE: Zone-to-zone interfaces join adjacent interfaces (those that share at least one tie line).

DeleteExisting: Set to YES or NO. YES means that previously defined interfaces will be deleted before the new interfaces are automatically inserted. NO means automatically inserted interfaces will be added to the list of previously defined interfaces. If this parameter is not specified, then YES is assumed.

UseDisplayFilters: Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting interfaces. NO specifies all Areas and Zones will be used to automatically insert interfaces regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

Prefix: An optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name. If this parameter is not specified, then "" is assumed.

Limits: Specifies the way the interfaces rating are set. Choices are:

AUTO: The interface rating is calculated based on the ratings of the components included in the interface.

ZEROS: All the interface limits are set to zero.

[value1, ..., value8]: The set of rating values to be used for the interfaces. The eight values must be specified, separated by commas.

The default parameter is ZEROS.

Script Run Mode Actions

The following script commands are among those available in Run Mode:

Animate(DoAnimate);

CalculatePTDF	([transactor seller],	[transactor buyer],	LinearMethod);
---------------	-----------------------	---------------------	----------------

[AREA num]	[AREA num]	AC
------------	------------	----

[ZONE num]	[ZONE num]	DC
------------	------------	----

[SUPERAREA "name"]	[SUPERAREA name]	DCPS
-----------------------	------------------	------

[INJECTIONGROUP "name"]	[INJECTIONGROUP name]
----------------------------	--------------------------

[BUS num]	[BUS num]
-----------	-----------

[SLACK]	[SLACK]
---------	---------

CalculatePTDFMultipleDirec tions	(StoreValuesForBranches,	StoreValuesForInterfaces,	LinearMethod);
-------------------------------------	--------------------------	---------------------------	----------------

YES or NO	YES or NO	AC
-----------	-----------	----

DC

DCPS

CalculateLODF	([BRANCH nearbusnum farbusnum ckt],	LinearMethod);
---------------	---	----------------

CalculateLODFMatrix	(WhichOnes,	filterProcess,	filterMonitor,	MonitorOnlyClosed,	LinearMethod)
---------------------	-------------	----------------	----------------	--------------------	---------------

OUTAGES	ALL	ALL	YES	DC
---------	-----	-----	-----	----

CLOSURES	SELECTED	SELECTED	NO	DCPS
----------	----------	----------	----	------

AREAZONE "filtername"	AREAZONE "filtername"
	SAME

CalculateTLR	([flowelement],	direction,	[transactor],	LinearMethod);
--------------	-----------------	------------	---------------	----------------

[INTERFACE "name"]	BUYER	same as above for PTDFs	AC
			DC

[BRANCH nearbusnum farbusnum ckt]	SELLER	DCPS
--------------------------------------	--------	------

CalculateTLRMultipleElement	(TypeElement	WhichElement,	direction,	[transactor],	LinearMethod
	BRANCH	SELECTED	BUYER	same as above for PTDFs	AC
	INTERFACE	OVERLOAD	SELLER		DC
	BOTH	CTGOVERLOAD			DCPS
CalculateVoltSense	([BUS num]);				
CalculateFlowSense	([flowelement], FlowType);				
	[INTERFACE "name"] MW				
	[BRANCH busnum1 busnum2 ckt] MVAR MVA				
CalculateLossSense	(FunctionType);				
	NONE				
	ISLAND				
	AREA				
	AREASA				
	SELECTED				
CalculateVoltToTransferSense	([transactor seller], [transactor buyer], TransferType TurnOffAVR);				
	[AREA num] [AREA num] [P, Q, or PQ] YES or NO				
	[ZONE num] [ZONE num]				
	[SUPERAREA "name"] [SUPERAREA name]				
	[INJECTIONGROUP "name"] [INJECTIONGROUP name]				
	[BUS num] [BUS num]				
	[SLACK] [SLACK]				
CalculateVoltSelfSense	(filter);				

SetSensitivitiesAtOutOfServiceToClosest;

ZeroOutMismatches;

Animate(DoAnimate);

Use this action to animate all the open oneline diagrams.

DoAnimate: Set to YES or NO. YES means to start the animation of the open oneline diagrams, while NO means that the animation will be paused.

CalculatePTDF([transactor seller], [transactor buyer], LinearMethod);

Use this action to calculate the PTDF values between a seller and a buyer. You may optionally specify the linear calculation method. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

[transactor seller]: The seller (or source) of power. There are six possible settings:

[AREA num], [AREA "name"], [AREA "label"]
[ZONE num], [ZONE "name"], [ZONE "label"]
[SUPERAREA "name"], [SUPERAREA "label"]
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
[BUS num], [BUS "name_nomkv"], [BUS "label"]
[SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

AC: Calculation including losses.
DC: Lossless DC.
DCPS: Lossless DC that takes into account phase shifter operation.

CalculatePTDFMultipleDirections(StoreValuesForBranches, StoreValuesForInterfaces, LinearMethod);

Use this action to calculate the PTDF values using multiple directions. Only directions with the field INCLUDE set to YES will be analyzed when calculating the multiple direction PTDFs. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

StoreValuesForBranches: Set to YES or NO. YES means that the PTDFs for lines/transformers will be stored.

StoreValuesForInterfaces: Set to YES or NO. YES means that the PTDFs for interfaces will be stored.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

AC: Calculation including losses.
DC: Lossless DC.
DCPS: Lossless DC that takes into account phase shifter operation.

CalculateLODF([BRANCH nearbusnum farbusnum ckt], LinearMethod);

Use this action to calculate the Line Outage Distribution Factors (or the Line Closure Distribution Factors) for a particular branch. If the branch is presently closed, then the LODF values will be calculated, otherwise the LCDF values will be calculated. You may optionally specify the linear calculation method as well. If no Linear Method is specified, Lossless DC will be used.

[BRANCH nearbusnum farbusnum ckt]: The branch whose status is being changed. Can also use strings

[BRANCH "nearbusname_kv" "farbusname_kv" ckt]

[BRANCH "nearbuslabel" "farbuslabel" ckt]

[BRANCH "label"]

LinearMethod: The linear method to be used for the LODF calculation. The options are:

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

Note: AC is NOT an option for the LODF calculation.

CalculateLODFMatrix(WhichOnes, filterProcess, filterMonitor, MonitorOnlyClosed, LinearMethod);

Use this action to calculate a matrix of LODF results comprised of multiple branches to be outaged or closed and multiple branches for which the impact of the outages/closures is determined.

WhichOnes: Specify the type of sensitivities to be calculated.

OUTAGES: Outage sensitivities will be calculated for those branches meeting the filterProcess.

CLOSURES: Closure sensitivities will be calculated for those branches meeting the filterProcess.

filterProcess: Specify a filter for the branches for which the outages or closures will be implemented.

ALL: All AC transmission lines.

SELECTED: Only those branches whose Selected field is set to *YES*.

AREAZONE: Only those branches meeting the area/zone filter.

"filtername": Name of advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

filterMonitor: Specify a filter for the branches for which the impact of the outages or closures will be determined.

ALL: All AC transmission lines.

SELECTED: Only those branches whose Selected field is set to *YES*.

AREAZONE: Only those branches meeting the area/zone filter.

"filtername": Name of advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

SAME: Same as set of branches to process as specified by filterProcess.

MonitorOnlyClosed: Set to YES to monitor only those branches that are closed. Set to NO to monitor branches regardless of their status.

LinearMethod: The linear method to use in the LODF calculations.

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

CalculateTLR([flow element], direction, [transactor], LinearMethod);

Use this action to calculate the TLR values a particular flow element (transmission line or interface). You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

[flow element]: This is the flow element we are interested in. Choices are:

[INTERFACE "name"]
[INTERFACE "label"]
[BRANCH nearbusnum farbusnum ckt]
[BRANCH "nearbusname_kv" "farbusname_kv" ckt]
[BRANCH "nearbuslabel" "farbuslabel" ckt]
[BRANCH "label"]

direction: the type of the transactor. Either BUYER or SELLER.

[transactor buyer]: the transactor of power. There are six possible settings.

[AREA num], [AREA "name"], [AREA "label"]
[ZONE num], [ZONE "name"], [ZONE "label"]
[SUPERAREA "name"], [SUPERAREA "label"]
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
[BUS num], [BUS "name_nomkv"], [BUS "label"]
[SLACK]

LinearMethod: The linear method to be used for the calculation. The options are:

AC: Calculation including losses.
DC: Lossless DC.
DCPS: Lossless DC that takes into account phase shifter operation.

CalculateTLRMultipleElement(TypeElement, WhichElement, direction, [transactor], LinearMethod);

Use this action to calculate the TLR values a multiple elements similar to as is done on the TLR multiple elements dialog. You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

TypeElement : May be either INTERFACE, BRANCH, or BOTH

WhichElement : There are three choices which represent which elements of the TypeElement specified will have TLR calculations performed.

SELECTED : Only branches or interfaces with their Selected Field = YES will be used.

OVERLOAD : Only branches that are presently overloaded using their normal ratings will be used

CTGOVERLOAD : You must have first run the contingency analysis. A branch or interface is included in the calculation if it has been overloaded during at least one contingency.

direction: the type of the transactor. Either BUYER or SELLER.

[transactor buyer]: the transactor of power. There are six possible settings.

[AREA num], [AREA "name"], [AREA "label"]
[ZONE num], [ZONE "name"], [ZONE "label"]
[SUPERAREA "name"], [SUPERAREA "label"]
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
[BUS num], [BUS "name_nomkv"], [BUS "label"]
[SLACK]

LinearMethod: The linear method to be used for the calculation. The options are:

AC: Calculation including losses.

DC: Lossless DC.

DCPS: Lossless DC that takes into account phase shifter operation.

CalculateVoltSense([BUS num]);

This calculates the sensitivity of a particular buses voltage to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

[BUS num]: The bus to calculate sensitivities for.

CalculateFlowSense([flow element], FlowType);

This calculates the sensitivity of the MW, MVAR, or MVA flow of a line or interface to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

[flow element]: This is the flow element we are interested in. Choices are:

[INTERFACE "name"]
[INTERFACE "label"]
[BRANCH busnum1 busnum2 ckt]
[BRANCH "name_kv1" "name_kv2" ckt]
[BRANCH "buslabel1" "buslabel2" ckt]
[BRANCH "label"]

FlowType: The type of flow to calculate this for. Either MW, MVAR, or MVA.

CalculateLossSense(FunctionType);

This action calculates the sensitivity of a real power loss function, PLosses, to bus real and reactive power injections. Stated mathematically, this action calculates $d \text{ PLosses} / d P_i$ and $d \text{ PLosses} / d Q_i$, where P_i and Q_i are the real and reactive power injections at bus i , respectively. Stated less formally, this action indicates how losses would change if one more MW or Mvar of power were injected at bus i .

FunctionType: Specifies how the losses are computed. Choices are:

NONE: No losses are calculated because a loss function is not specified.

ISLAND: Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

AREA: Losses are calculated with respect to the total losses for the area containing bus i. This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

AREASA: Losses are calculated with respect to the total losses for the area containing bus i if bus i does not belong to a super area, and with respect to the total losses for the super area containing bus i if bus i does belong to a super area.

SELECTED: Losses are calculated with respect to the total losses for a group of areas whose *selected* field is set to YES.

CalculateVoltToTransferSense([transactor seller], [transactor buyer], TransferType, TurnOffAVR);

This calculates the sensitivity of bus voltage to a real or reactive power transfer between a seller and a buyer. The sensitivity is calculated for all buses in the system.

[transactor seller]: This is the seller (or source) of power. There are six possible settings:

[AREA num], [AREA "name"], [AREA "label"]
[ZONE num], [ZONE "name"], [ZONE "label"]
[SUPERAREA "name"], [SUPERAREA "label"]
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
[BUS num], [BUS "name_nomkv"], [BUS "label"]
[SLACK]

[transactor buyer]: This is the buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

TransferType: The type of power transfer. The options are:

P – real power transfer
Q – reactive power transfer
PQ – both real and reactive power transfer. (Note: Real and reactive power transfers are calculated independently, but both are calculated.)

TurnOffAVR: Set to YES or NO. Set to YES to turn off AVR control for generators participating in the transfer. Set to NO to leave the AVR control unchanged for generators participating in the transfer.

CalculateVoltSelfSense(filter);

This calculates the sensitivity of a particular bus' voltage to real and reactive power injections at the same bus. (Note: This assumes that the power is injected at a given bus and taken out at the slack bus.)

filter: There are four options for the filter:

CalculateVoltSelfSense; : No filter specified means to merge all the buses of the system.

CalculateVoltSelfSense("filtername"); : "filtername" means to merge those buses that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

CalculateVoltSelfSense(AREAZONE); : AREAZONE means to merge those buses that meet the area/zone filters.

CalculateVoltSelfSense(SELECTED); : SELECTED means to merge those buses whose *selected* field is set to YES.

SetSensitivitiesAtOutOfServiceToClosest;

This will take the P Sensitivity and Q Sensitivity values calculated using CalculateTLR, CalculateFlowSense, or CalculateVoltSense actions and then populate the respective values at out-of-service buses so that they are equal to the value at the closest in service bus. The "distance" to the in-service buses will be measured by the number of nodes. If an out-of-service bus is equally close to a set of buses, then the average of that set of buses will be used.

ZeroOutMismatches;

With this script command, the bus shunts are changed at each bus that has a mismatch greater than the MVA convergence tolerance so that the mismatch at that bus is forced to zero.

In addition to these general run mode actions, there are actions related to specific types of analysis: [PowerFlow](#), [Contingency](#), [ATC](#), [Fault](#), [PV](#) and [QV](#).

Script PowerFlow Related Actions

The following actions are available in Run Mode:

```
DoCTGAction([contingency action]);  
SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);  
ResetToFlatStart(FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);  
SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);  
InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);  
SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);  
SolveFullSCOPF(BCMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);  
OPFWriteResultsAndOptions ("filename");  
DiffFlowSetAsBase;  
DiffFlowClearBase;  
DiffFlowMode (diffmode);  
PRESENT  
BASE  
DIFFERENCE
```

DoCTGAction([contingency action]);

Call this action to use the formats seen in the CTGElement subdata record for Contingency Data. Note that all actions are supported, except COMPENSATION sections are not allowed.

SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a single power flow solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

SolMethod The solution method to be used for the power flow calculation. The options are:

RECTNEWT: For Rectangular Newton-Raphson method.

POLARNEWTON: For Polar Newton-Raphson method.

GAUSSSEIDEL: For Gauss-Seidel method.

FASTDEC: For Fast Decoupled method.

DC: For DC method.

ROBUST: For ROBUST method.

The default method is RECTNEWT.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

ResetToFlatStart (FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);

Use this action to initialize the Power Flow Solution to a "flat start." The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

FlatVoltagesAngles Set to YES or NO. YES means setting all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero. Default Value is YES.

ShuntsToMax Set to YES or NO. YES means to increase Switched Shunts Mvar half way to maximum. Default Value is NO.

LTCsToMiddle Set to YES or NO. YES means setting the LTC Transformer Taps to middle of range. Default Value is NO.

PSAnglesToMiddle Set to YES or NO. YES means setting Phase Shifter angles to middle of range. Default Value is NO.

SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a primal LP OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

This action is basically identical to the SolvePrimalLP action, except that this will only perform a single optimization. The SolvePrimalLP will iterate between solving the power flow and an optimization until this iteration converges. This action will only solve the optimization routine once, then resolve the power flow once and then stop.

InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

This command clears all the structures and results of previous primal LP OPF solutions. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

SolveFullSCOPF (BCMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);

Call this action to perform a full Security Constrained OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

BCMethod The solution method to be used for solving the base case. The options are:

POWERFLOW – for single power flow algorithm.

OPF – for the optimal power flow algorithm.

Default Value is POWERFLOW.

"filename1" The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

"filename2" The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

CreateIfNotFound1 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

CreateIfNotFound2 Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

OPFWriteResultsAndOptions("filename");

Writes out all information related to OPF analysis as an auxiliary file. This includes Limit Monitoring Settings, options for Areas, Buses, Branches, Interfaces, Generators, SuperAreas, OPF Solution Options.

DiffFlowSetAsBase;

Call this action to set the present case as the base case for the difference flows abilities of Simulator.

DiffFlowClearBase;

Call this action to clear the base case for the difference flows abilities of Simulator.

DiffFlowMode(diffmode);

Call this action to change the mode for the difference flows abilities of Simulator.

diffmode:

String that starts with 'P' changes it to PRESENT

String that starts with 'B' changes it to BASE

String that starts with 'D' changes it to DIFFERENCE

Script Contingency Related Actions

The following actions are available in Run Mode:

```
CTGSolveAll;  
CTGSolve("ContingencyName");  
CTGSetAsReference;  
CTGRestoreReference;  
CTGProduceReport("filename");  
CTGWriteResultsAndOptions("filename", [opt1, opt2, opt3, ... opt9], KeyField, UseDATASection);  
CTGWriteFilePTI("filename", BusFormat, TruncateCTGLabels);  
CTGAutoInsert;  
CTGConvertAllToDeviceCTG;  
CTGCalculateOTDF( [transactor seller], [transactor buyer], LinearMethod);  
[AREA num] [AREA num] AC  
[ZONE num] [ZONE num] DC  
[SUPERAREA "name"] [SUPERAREA "name"] DCPS  
[INJECTIONGROUP "name"] [INJECTIONGROUP "name"]  
[BUS num] [BUS num]  
[SLACK] [SLACK]
```

CTGSolveAll;

Call this action to solve all the contingencies which are not marked skip.

CTGSolve("ContingencyName");

Call this action solve a particular contingency. The contingency is denoted by the "Contingency Name".

CTGSetAsReference;

Call this action to set the present system state as the reference for contingency analysis.

CTGRestoreReference;

Call this action to reset the system state to the reference state for contingency analysis.

CTGProduceReport("filename");

Produces a text-based contingency analysis report using the settings defined in CTG_Options.

CTGWriteResultsAndOptions("filename", [opt1, opt2, opt3, ... opt9], KeyField, UseDATASection);

Writes out information related to contingency analysis as an auxiliary file. The information included in the auxiliary file depends on the options settings described below. Each of the nine options settings parameters is either a YES or NO entry. If the options settings are not specified or blank, the default entry given for each will be used.

"filename": name of AUX file to save

Opt1 - Save unlinked contingency actions, default = NO

Opt2 - Save contingency options, default = YES

Opt3 - Save limit monitoring settings, default = NO

Opt4 - Save general power flow solution options, default = YES

Opt5 - Save list display settings, default = NO

Opt6 - Save contingency results, default = YES

Opt7 - Save inactive violations, default = YES

Opt8 - Save interface definitions, default = NO

Opt9 - Save injection group definitions, default = NO

KeyField: indicates the identifier that should be used for the data. Valid entries are PRIMARY, SECONDARY, or LABEL. The default setting is PRIMARY. PRIMARY will save using the bus numbers and other primary key fields. SECONDARY will save using bus name and nominal kV and other secondary fields. LABEL will save using device labels. If no labels are specified then the primary key field will be used.

UseDATASection: set this to YES or NO. If set to YES, data that by default is specified using SUBDATA sections will instead be specified using DATA sections. For example, the actions that define a contingency by default are specified using a SUBDATA section. If choosing to use the DATA section instead, each action will be specified in a DATA record belonging to the ContingencyElement objecttype.

CTGWriteFilePTI("filename", BusFormat, TruncateCTGLabels);

Writes the contingency definitions to a PTI CON text file format.

"filename": name of the text file to save

BusFormat: the entries for the bus format are NUMBER, NAME8 or NAME12, which indicate to save the contingency definitions using bus numbers, eight character bus names, or twelve character bus names, respectively

TruncateCTGLabels: the entries for this field is either YES or NO, depending on if you would like the output to truncate the contingency names to twelve characters or leave them as defined if longer than twelve characters in Simulator

CTGAutoInsert;

Automatically generates a contingency list using the settings defined in CTG_Options.

CTGConvertAllToDeviceCTG;

This command is intended for use with full topology models, where breakers and disconnects are defined in addition to generators, loads, transmission lines, and so on. This function would have no effect on a traditional planning model representation, which has no breakers or disconnects explicitly defined.

The purpose of the function is to allow the user to take a contingency set that is defined with outages of breakers and disconnects in a full topology model and convert them to outages of the traditional planning model elements, such as generators, loads, transmission lines, etc. This would be used in conjunction with the ability to save a full topology model as a consolidated model. A consolidated model reduces the full model down to a traditional planning model by examining the breaker and disconnect statuses, and reducing the system down by consolidating breakers and disconnects that are in service. The resulting model is a smaller model with the traditional planning elements represented, but breakers and disconnects have been removed and nodes aggregated into bus representations. This function will also take the breaker and disconnect statuses and convert contingencies defined with the breakers and disconnects and convert them into contingencies of the planning model devices affected by opening the original breakers or disconnects. Thus you could create a contingency set that is defined for the consolidated model, and can be run on the consolidated model with the same results as if the original contingency set is run on the full topology model.

Note that the contingency set generated depends on the statuses of the breakers and disconnects, and that the contingencies created will be different for different statuses of breakers and disconnects in the full topology model.

CTGCalculateOTDF([transactor seller], [transactor buyer], LinearMethod);

This action calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis.

The parameters will be used to calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

NOTE: All the parameters are optional. If they are not specified, then the existing PTDFs will be used.

[transactor seller]: The seller (or source) of power. There are six possible settings:

- [AREA num]
- [ZONE num]
- [SUPERAREA "name"]
- [INJECTIONGROUP "name"]
- [BUS num]
- [SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

LinearMethod: The linear method to be used for the PTDF calculation. The options are:

- AC: for calculation including losses.
- DC: for lossless DC.
- DCPS: for lossless DC that takes into account phase shifter operation.

Script ATC Related Actions

The following actions are available in Run Mode:

ATCDetermine	([transactor seller],	[transactor buyer],	DoDistributed);
	[AREA num]	[AREA num]	YES
	[ZONE num]	[ZONE num]	NO
	[SUPERAREA "name"]	[SUPERAREA name]	
	[INJECTIONGROUP "name"]	[INJECTIONGROUP name]	
	[BUS num]	[BUS num]	
	[SLACK]	SLACK]	


```

ATCRestoreInitialState;
ATCIncreaseTransferBy      (amount);
ATCTakeMeToScenario      (RL,                G,                I)
                           ;
ATCDetermineATCFor      (RL,                G,                I,      ApplyTransfer)
                           ;
ATCWriteResultsAndOptions ("filename",      AppendFile)
                           ;
ATCWriteToExcel          ("worksheetname")
                           ;
ATCWriteToText           ("filename",      filetype);
  
```

ATCDetermine([transactor seller], [transactor buyer], DoDistributed);

Use this action to calculate the Available Transfer Capability (ATC) between a seller and a buyer. Note that the buyer and seller must not be same thing. Other options regarding ATC calculations should be set using a DATA section via the ATC_Options object type.

[transactor seller]: The seller (or source) of power. There are six possible settings

[AREA num], [AREA "name"], [AREA "label"]
 [ZONE num], [ZONE "name"], [ZONE "label"]
 [SUPERAREA "name"], [SUPERAREA "label"]
 [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
 [BUS num], [BUS "name_nomkv"], [BUS "label"]
 [SLACK]

[transactor buyer]: The buyer (or sink) of power. There are six possible settings which are the same as for the seller.

DoDistributed: Set to YES to use the distributed ATC solution method.

ATCRestoreInitialState;

Call this action to restore the initial state for the ATC tool.

ATCIncreaseTransferBy(amount);

Call this action to increase the transfer between the buyer and seller.

ATCTakeMeToScenario(RL, G, I);

Call this action to set the present case according to Scenario RL, G, I.

ATCDetermineATCFor(RL, G, I, ApplyTransfer);

Call this action to determine the ATC for Scenario RL, G, I.

ApplyTransfer: Set this value to YES to leave the system state at the transfer level that was determined. When using the Iterated Linear then Full Contingency solution method, the system state will retain the transfer level but the contingency will not be applied.

ATCWriteResultsAndOptions("filename",AppendFile);

Writes out all information related to ATC analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Solution Options, DataGrid settings relevant to ATC, ATC Options, ATC results, as well as any Model Criteria that are used by the Contingency Definitions.

"filename": Name of the auxiliary file.

AppendFile: This is an optional parameter. YES means to append results to existing "filename". NO means to overwrite "filename" with results. Default is YES.

ATCWriteToExcel("worksheetname");

Sends ATC analysis results to an Excel spreadsheet. This script command is available only for Multiple Scenarios ATC analysis.

"worksheetname": The name of the Excel sheet where the results will be sent.

ATCWriteToText("filename",filetype);

This is used with Multiple Scenario ATC analysis. Multiple files are created with "filename" as the primary identifier and the Interface scenario label appended to the end of the filename. Separate files are created for each of the Interface scenarios. Results inside the files are separated into sections based on the number of Rating/Load scenarios.

"filename": Primary identifier for the name of the file in which to save the results. "filename" gets appended with the Interface scenario label to complete the filename.

filetype: Either TAB or CSV. This indicates the delimiter to use when writing out the file(s). This is an optional parameter with TAB being the default if omitted.

Script Fault Related Actions

The following actions are available in Run Mode:

Fault	([BUS num],		faulttype,	R,	X);
Fault	([BRANCH nearbusnum farbusnum ckt],	faultlocation,	faulttype,	R,	X);
					;
			SLG		
			LL		
			3PB		
			DLG		

Fault([Bus num, faulttype, R, X]);

Fault([BRANCH nearbusnum farbusnum ckt], faultlocation, faulttype, R, X);

Call this function to calculate the fault currents for a fault. If the fault element is a bus then do not specify the fault location parameter. If the fault element is a branch, then the fault location is required.

[Bus num]: This specifies the bus at which the fault occurs. You may also specify the bus using secondary keys or labels:

[BUS "name_nomkv"]

[BUS "label"]

[BRANCH nearbusnum farbusnum ckt]: This specifies the branch on which the fault occurs. You may also specify the branch using secondary keys or labels:

[BRANCH "name_kv1" "name_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

Fault location: This specifies the percentage distance along the branch where the fault occurs. This percent varies from 0 (meaning at the nearbus) to 100 (meaning at the far bus)

Faulttype: This specified the type of fault which occurs. There are four options:

SLG: Single Line To Ground fault

LL: Line to Line Fault

3PB: Three Phase Balanced Fault

DLG: Double Line to Group Fault.

R, X: These parameters are optional and specify the fault impedance. If none are specified, then a fault impedance of zero is assumed.

Script PV Related Actions

The following actions are available Run Mode:

```
PVSetSourceAndSink      ([elementSource],      [elementSink]);
                        [INJECTIONGROUP    [INJECTIONGROUP
                        "name"]            "name"]

PVRun;

PVRun      ([elementSource],      [elementSink]);
            [INJECTIONGROUP "name"] [INJECTIONGROUP "name"]

PVClear;
PVStartOver;
PVDestroy;

PVWriteResultsAndOptions ("filename", AppendFile);

RefineModel  (objecttype,  filter,  Action          ForWhichValue);
              AREA        TRANSFORMERTAPS
              ZONE        SHUNTS
                       OFFAVR
```

Changes were made with Simulator version 14 to eliminate the need for a PV study name. To maintain functionality with any existing processes that users might have in place using older script definitions, scripts from older versions of Simulator will still be supported if the name is specified. However, the name will just be ignored. The script formats given here reflect the changes for version 14.

The PVCreate script required in previous versions is no longer necessary in Simulator version 14. Version 14 will still recognize this action if it is included and will simply set the source and sink for the study. This does the same thing as PVSetSourceAndSink.

It is highly recommended that for any new processes the new script formats specified here be used.

PVSetSourceAndSink([elementSource], [elementSink]);

Call the function to specify the source and sink elements to perform the PV study.

[element source]: The source of power for the PV study. There is only one possible setting:
[INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

[element sink]: The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

PVRun;

Call the function to start the PV study.

PVRun([elementSource], [elementSink]);

Call the function to specify the source and sink elements and start the PV study.

[element source]: The source of power for the PV study. There is only one possible setting:

[INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

[element sink]: The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

PVClear;

Call the function to clear all the results of the PV study.

PVStartOver;

Call the function to start over the PV study. This includes clear the activity log, clear results, restore the initial state, set the current state as initial state, and initialize the step size.

PVDestroy;

Call the function to destroy the PV study. This will remove all results and prevent any restoration of the initial state that is stored with the PV study.

PVWriteResultsAndOptions("filename", AppendFile);

Call this action to save all the PV results and options in the auxiliary file "filename".

AppendFile YES/NO optional parameter. Specifying NO will cause it to overwrite the existing file. Specifying YES will cause it to append to the file. If not specified, then YES is assumed.

RefineModel(objecttype, filter, Action, Tolerance);

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

objecttype: The objecttype being selected.

AREA

ZONE

filter: There are three options for the filter:

RefineModel(..., "", ...);: No filter specified means to select all objects of this type.

RefineModel(..., "filtername", ...);: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Action: The way the model will be refined. Choices are:

TRANSFORMERTAPS Fix all transformer taps at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their Qmax – Qmin is less or equal to the user specified tolerance.

Tolerance: Tolerance value.

Script QV Related Actions

The following actions are available in Run Mode:

QVRun ("filename", InErrorMakeBaseSolvable);
YES or NO

NOTE: The QV study is always performed on selected buses.

QVWriteResultsAndOptions ("filename");

RefineModel (objecttype, filter, Action ForWhichValue);
AREA TRANSFORMERTAPS
ZONE SHUNTS
OFFAVR

QVRun("filename", InErrorMakeBaseSolvable);

Call the function to start a QV study for the list of buses whose SELECTED? field is set to YES.

"filename": This specifies the file to which to save a comma-delimited version of the results.

InErrorMakeBaseSolvable: This specifies whether to perform a solvability analysis of the base case if the pre-contingency base case can not be solved. If not specified, then YES is assumed.

QVWriteResultsAndOptions("filename");

Call this action to save all the QV results and options in the auxiliary file "filename".

RefineModel(objecttype, filter, Action, Tolerance);

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

objecttype: The objecttype being selected.

AREA

ZONE

filter: There are three options for the filter:

RefineModel(..., "", ...);: No filter specified means to select all objects of this type.

RefineModel(..., "filtername", ...);: "filtername" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filtername" then takes on the form "<objecttype>objecttype's filtername".

Action: The way the model will be refined. Choices are:

TRANSFORMERTAPS Fix all transformer taps at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their $Q_{max} - Q_{min}$ is less or equal to the user specified tolerance.

Tolerance: Tolerance value.

Script Topology Processing Related Actions

```
CloseWithBreakers (objecttype, "filename",  
OnlyEnergizeSpecifiedObjects);
```

The following action is available in Run Mode:

```
SaveConsolidatedCase ("filename" filetype,  
, TruncateCtgLabel AddCommentsForObjectLabels]  
[BusFormat s, ];  
,
```

CloseWithBreakers(objecttype,"filename",OnlyEnergizeSpecifiedObjects);

This action is used to specify which devices can be energized by closing breakers.

objecttype Objects that are valid to energize. Only allowed for Gen, Load Buses, Transmission Lines, Switched Shunts, DC Lines, Bus Injection Groups and Interfaces.

filename "filename" means select those that meet the advanced filter. Advanced filters belonging to a different objecttype can also be used. The "filename" then takes on the form "<objecttype>objecttype's filename".

OnlyEnergizeSpecifiedObjects optional parameter, default is NO.

Yes –No extra objects in addition to those specified in the filter can be energized. Each object will be evaluated individually.

No– Extra objects could be energized in addition to those specified if a group of breakers required to energize a specified object also causes other objects to be energized. All objects will be evaluated collectively for determining which objects can be energized, i.e. breakers that cause one object to be energized might also be needed for another object to be energized.

See the [Close Breakers Overview](#) topic for more information on how the close with breakers process works.

SaveConsolidatedCase("filename", filetype, [BusFormat, TruncateCtgLabels, AddCommentsForObjectLabels]);

This action saves the full topology model into a consolidated case.

"filename" : The name of the consolidated case file to be saved.

Filetype : Optional parameter to specify the type of the file to be saved. If omitted, the latest version of the PWB will be used.

PWB – save a pwb file with the most recent version

PWBX – save a pwb with version X

PWPTI – save file with most recent PTI file format. When contingencies are present, they are saved to a .CON file.

PWPTIXX – save the file with PTI version XX, where XX is between 23 and 32

BusFormat : optional parameter used to specify the bus identifier format in the .CON file used to store contingencies when saving a PTI file

Number – identify buses using number

Name8 – identify buses using the Name_kV identifier truncated to 8 characters

Name12 – identify buses using the Name_kV identifier truncated to 12 characters

TruncateCTGLabels : optional parameter used to specify if the contingency labels should be truncated to 12 characters when saving the contingencies in PTI format

YES – truncate the contingency labels to 12 characters

NO – do not truncate the contingency labels

AddCommentsForObjectLabels : (optional) YES adds object labels to the end of data records when saving a RAW file. (default NO)

Script Transient Stability Related Actions

The following actions are available in Run Mode:

TSSolveAll;			
TSSolve	(ContingencyName	[StartTime,	StopTime,
	, StepSize,	StepInCycles]);	
TSSolveOptions	("FileName",	[SaveDynamicModel,	SaveStabilityOptions,
	SaveStabilityEvents	SaveResultsSettings,	SavePlotDefinitions],
	, Keyfield]);		
TSSaveTwoBusEquivalent	("PwbFileName",	[BUS]);	
TSCalculateSMIBEigenValues;			
TSLoadGE	("FileName",	GENCCYN);	
TSLoadPTI	("FileName",	"MCREfilename",	"GNETfilename",
	"BASEGENfilename		
	");		
TSLoadBPA	("FileName");		
TSGetResults	("FileName",	Single/Separate,	[Contingencies,
	[Plots,	StartTime,	EndTime);
	ObjectFields],		

TSSolveAll;

Solves all defined transient contingencies which are not set to skip.

TSSolve("ContingencyName", [StartTime, StopTime, StepSize, StepInCycles]);

Solves only the specified contingency.

ContingencyName : The name of the contingency to solve

StartTime : (optional) Start time in seconds

StopTime : (optional) Stop time in seconds

StepSize : (optional) Step size in seconds

Example usage(s) :

```
TSSolve("ctg test");
TSSolve(ctg test);
TSSolve("test2");
TSSolve(test2);
TSSolve(test2, [0.0, 10, 0.5, Yes]);
TSSolve(test2, [1.0, 9, 0.5, Yes]);
TSSolve("test2", [1.0, 9, 0.01, No]);
```

TSWriteOptions("FileName", [SaveDynamicModel, SaveStabilityOptions, SaveStabilityEvents, SaveResultsEvents, SavePlotDefinitions], KeyField);

Save the transient stability option settings to an auxiliary file.

FileName : Name and path of the file to save

SaveDynamicModel : (optional) NO doesn't save dynamic model (default YES)

SaveStabilityOptions : (optional) NO doesn't save stability options (default YES)

SaveStabilityEvents : (optional) NO doesn't save stability events (default YES)

SaveResultsSettings : (optional) NO doesn't save results settings (default YES)

SavePlotDefinitions : (optional) NO doesn't save plot definitions (default YES)

KeyField : (optional) Specifies key: can be Primary, Secondary, or Label (default Primary)

Example usage(s) :

```
TSWriteOptions(auxtest.aux);
TSWriteOptions("auxtest.aux");
TSWriteOptions("auxtest.aux", [YES, NO, NO, NO, NO]);
TSWriteOptions("auxtest.aux", [YES, NO, "", NO, NO, P]);
TSWriteOptions("auxtest.aux", [YES, NO, NO, "", YES, S]);
```

TSSaveTwoBusEquivalent ("PwbFileName", [BUS]);

Save the two bus equivalent model of a specified bus to a PWB file. Initialization to the start time is always done before saving the two bus equivalent.

PwbFileName : Name and path for the output PWB file

BUS : Bus can be specified in three ways:

Number : [BUS busnum]

Name/NomkV : [BUS "busname_nominalKV"]

Label : [BUS "buslabel"]

Example usage(s) :

```
TSSaveTwoBusEquivalent("TwoBusTest.pwb",[Bus 2]);
```

TSCalculateSMIBEigenValues;

Calculate single machine infinite bus eigenvalues. Initialization to the start time is always done before calculating eigenvalues.

To save out the SMIB results to a file, use the SaveData script function after the eigenvalues have been calculated.

Example usage(s) :

```
TSCalculateSMIBEigenValues;
SaveData("C:\auxTest.aux", AUXDEF, PWTXGEN, [BusNum, BusName, GenID, AreaName,
TSGenMachineName, TSGenExciterName, TSGenGovernorName, TSGenStabilizerName,
TSEigenValueValid, TSNuEigenValue, TSNuEigenValueZero, TSEigenValueMax,
TSEigenValueMin, TSComplexEigenValue, TSComplexEigenValue:1, TSComplexEigenValue:2], []);
```

TSLoadGE("FileName", GENCCYN);

Loads transient stability data stored in the GE DYD format.

FileName : Name of the DYD file to load

GENCCYN : YES to split combined cycle units, NO to leave them alone

TSLoadPTI("FileName", "MCREfilename", "MTRLOfilename", "GNETfilename", "BASEGENfilename");

Loads transient stability data in the PTI format.

FileName : Name of the DYR file to load

MCREfilename : (optional) If not loading a MCRE file, specify ""

MTRLOfilename : (optional) If not loading a MTRLO file, specify ""

GNETfilename : (optional) If not loading a GNET file, specify ""

BASEGENfilename : (optional) If not loading a BASEGEN file, specify ""

TSLoadBPA("FileName");

Loads transient stability data stored in the BPA format.

FileName : Name of the BPA file to load

**TSGetResults("FileName", Single/Separate, [Contingencies], [Plots, ObjectFields],
StartTime, EndTime);**

Use this to save out results for specific variables from plots, subplots, and object/field pairs after a transient stability simulation has been run. If StartTime and StopTime are not specified, results for the entire simulation time are obtained.

FileName : Name of the CSV result file to write out

Single/Separate : Determines whether the results are all saved in one file (Single) with name "filename_Results.csv" or whether results for each transient contingency is saved in a separate file (Separate) with name "filename_ctgname.csv." A separate header file is also saved out, with a name of "filename_Header.csv".

Contingencies : A list of contingency names for which to save out results

Plots, ObjectFields : A list of plots and object/field pairs to save out for the specified contingencies

StartTime : Start of the window of simulation time from which the results are to be retrieved

EndTime : End of the window of simulation time from which the results are to be retrieved

Example usage(s) :

```
TSGetResults("filename.csv", SEPARATE, [ctg test], ["Plot 'Gen_Rotor Angle", "Bus 4 |  
frequency"]);
```

```
TSGetResults("filename.csv", SEPARATE, [test2, test3], ["Plot 'Gen_Rotor Angle"], 0.0, 10.0);
```

```
TSGetResults("filename.csv", SEPARATE, ["test3", ctg test], ["Plot 'Gen_Rotor Angle"], 0.0,  
10.0);
```

```
TSGetResults("filename.csv", SINGLE, ["test2"], ["Plot 'Gen_Rotor Angle"], 0.0, 10.0);
```

Data Section

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

DATA DataName (object_type, [list_of_fields], file_type_specifier, create_if_not_found)

```
{  
data_list_1  
.  
.  
.  
data_list_n  
}
```

Immediately following the DATA keyword, you may optionally include a DataName. By including the DataName, you can make use of the script command LoadData("filename", DataName) to call this particular data section from another auxiliary file.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).

Data Argument List

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" and ")" mark the beginning and end of the argument list.

The **file_type_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file_type_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **create_if_not_found** parameter allows the specification of whether or not one or more data records that exist in the data section that do not link to existing data in the case be created when read from the auxiliary file. The parameter is optional, and if omitted Simulator will prompt the user regarding creating objects not found in the case from records in the data section. If the parameter is specified as True, then objects not found will be created from the data records, and if False objects not found will not be created from the data records. Data records for which objects already exist in the model will be read and applied regardless of this parameter setting.

The **object_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object_type** equals BUS, then the data describes BUS objects. Below is a list of many of the object types recognized by Simulator:

3WXFormer	DefDrawSuperArea	OPF_Options_and_Results
Area	DefDrawZone	OPFSolutionSummary
ATC_Options	Direction	Owner
ATCExtraMonitor	DynamicFormatting	PieSizeColorOptions
ATCGeneratorChange	Equiv_Options	PostPowerFlowActions
ATCInterfaceChange	ExportBidCurve	PTDF_Options
ATCLineChange	Filter	PVCurve_Options
ATCLineChangeB	GEMotor	PWCaseInformation
ATCScenario	Gen	PWFormOptions
ATCZoneChange	GeoDataViewStyle	PWLPOPFGTGViol
AuxFileExportFormatData	GlobalContingencyActions	PWLPTabRow
AuxFileExportFormatDisplay	GScenarioName	PWOPFTimePoint
BGCalculatedField	HintDefValues	PWPVResultListContainer
Branch	IG_AutoInsert_Options	QVCurve
Bus	ImportBidCurve	QVCurve_Options
BusGroupSwapRec	ImportExportBidCurve	RLScenarioName
BusNumberSwap	InjectionGroup	Scale_Options
BusViewFormOptions	Interface	Schedule
CaseInfo_Options	InterfaceAElement	ScreenLayer
ColorMap	InterfaceBElement	SelectByCriteriaSet
Contingency	IScenarioName	ShapefileExportDescription
Ctg_AutoInsert_Options	Limit_Monitoring_Options	ShortCut

Ctg_Options	LimitCost	Shunt
CTGElementBlock	LimitSet	Sim_Environment_Options
CustomCaseInfo	Load	Sim_Misc_Options
CustomCaseInfoRow	LODF_Options	SimSimulation_Options
CustomColors	LPOPFMarginalControls	Sim_Solution_Options
CustomExpression	LPVariable	StudyMWTransactions
DataGrid	MessLog_Options	Substation
DCTransmissionLine	ModelCondition	SuperArea
DefDrawArea	ModelExpression	TLR_Options
DefDrawBackground	ModelFilter	TransferLimiter
DefDrawBus	MTDCRecord	Transformer
DefDrawGen	MultiSectionLine	TSSchedule
DefDrawInterface	MvarMarginalCostValues	UC_Options
DefDrawLineTransformer	MWMarginalCostValues	ViolationCTG
DefDrawLoad	MWTransaction	WhatOccurredDuringContingency
DefDrawShunt	Nomogram	XFCorrection
DefDrawSubstation	NomogramInterface	Zone

The list of object types Simulator's auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**.

The **list_of_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list_of_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 3,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list_of_fields** are:

- The **list_of_fields** may take up several lines of the text file.
- The **list_of_fields** should be encompassed by braces [].
- When encountering the PowerWorld comment string `'''` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'''` will be ignored as comments.
- Field names must be separated by commas.

Example DATA Argument List

```
DATA (BUS, [BusNomKV, Bus, // comment here
ABCPhaseAngle: 1, ABCPhaseAngle: 2, ABCPhaseV, ABCPhaseV: 1,
// comments allowed here to
```

```
// note that blank rows are ignored
AreaNum, BusAngle, BusB, BusCat, BusEquiv, BusG,
BusGenericSensV, BusKVVolt, BusLambda, BusLoadMVA, // more comment
BusLoadMW, BusLongName],True)
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field LineMW. For a branch, there are two MW flows associated with the line: one MW flow at the from bus, and one MW flow at the to bus. So that the number of fields does not become huge, the same field name is used for both of these values. For the from bus flow, we write LineMW: 0, and for the to bus flow, we write LineMW: 1. Note that fieldnames such as LineMW: 0 may simply leave off the: 0.

The **create_if_not_found** field is optional. This specifies whether or not to create a new object if an existing one is not found. If the value is YES, objects will be created. If the value is NO, objects will not be created. If omitted, the default behavior of prompting the user about whether or not to create a non-existing object will continue. If loading and auxiliary file using the **LoadAux** script command, the **create_if_not_found** field for the data section will override the **CreateIfNotFound** field with the script.

Data Key Fields

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for BUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for GEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list_of_fields** (i.e. they need not be the first fields listed in **list_of_fields**). As long as the key fields are present, Simulator can identify the specific object. By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Case Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks *.

Data List

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list_of_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value_list**:

- The **value_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data_list**.
- If the **file_type_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.
- A string in the format `&[ModelExpressionName]:[#digits]:[#decimals]` will be treated as though the value of the named model expression is entered in the field, with `[#digits]` total and `[#decimals]` digits to the right of the decimal point. This format may be used in case information displays as well as in AUX files.

Data SubData Sections

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<SubData>`, `</SubData>` tags that store lists of information about a particular object. This formatting looks like the following:

DATA (object_type, [list_of_fields], file_type_specifier)

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
.
.
.
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.

Data ATC_Options

RLScenarioName

GScenarioName

IScenarioName

These three sections contain the pretty names of the RL Scenarios, G Scenarios, and I Scenarios. Each line consists of two values: Scenario Number and a name string enclosed in quotes.

Scenario Number: The scenarios are number 0 through the number of scenarios minus 1.

Scenario Name: These represent the names of the various scenarios.

Example:

```
<SUBDATA RLScenarioName>
//Index Name
0 "Scenario Name 0"
1 "Scenario Name 1"
</SUBDATA>
```

ATCMemo

This section contains the memo text for the ATC analysis.

Example:

```
<SUBDATA ATCMemo>
//Memo
"Comments for the ATC analysis"
</SUBDATA>
```

Data ATCExtraMonitor

ATCFlowValue

This subdata section contains a list of a flow values for specified transfer levels. Each line consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW).

Flow Value: Contains a string describing which monitor this belongs to.

Transfer Level: Contains the value for this extra monitor at the last linear iteration.

Example:

```
<SUBDATA ATCFlowValue>
//MWFlow TransferLevel
94.05 55.30
105.18 80.58
109.02 107.76
</SUBDATA>
```

Data ATCScenario

TransferLimiter

This subdata section contains a list of the TransferLimiters for this scenario. Each line contains fields relating one of the Transferlimiters. The fields are written out in the following order:

Limiting Element: Contains a description of the limiting element. The possible values are

"PowerFlow Divergence"

"AREA num"

"SUPERAREA name"

"ZONE num"

"BRANCH num1 num2 ckt"

"INJECTIONGROUP name"

"INTERFACE name"

Limiting Contingency: The name of the limiting contingency. If blank, then this means it's a limitation in the base case.

MaxFlow: The transfer limitation in MW.

PTDF: The PTDF on the limiting element in the base case (not in percent).

OTDF: The OTDF on the limiting element under the limiting contingency.

LimitUsed: The limit which was used to determine the MaxFlow in per unit.

PreTransEst: The estimated flow on the line after the contingency but before the transfer in per unit.

MaxFlowAtLastIteration: The total transfer at the last iteration in per unit.

IterativelyFound: Either YES or NO depending on whether it was iteratively determined.

Example:

```
<SUBDATA TransferLimiter >
"BRANCH 40767 42103 1" "Contin" 2.84 -0.0771 -0.3883 -4.35 -4.35 -0.01 "-55.88" YES
"BRANCH 42100 42321 1" "Contin" 4.42 0.1078 0.5466 6.50 5.64 1.57 " 22.59" NO
"BRANCH 42168 42174 1" "Contin" 7.45 -0.0131 -0.0651 -1.39 -1.09 4.60 "-33.31" NO
"BRANCH 42168 42170 1" "Contin" 8.54 0.0131 0.0651 1.39 1.02 5.69 " 26.10" NO
"BRANCH 41004 49963 1" "Contin" 9.17 -0.0500 -0.1940 -4.39 -3.16 6.32 " 68.73" NO
"BRANCH 46403 49963 1" "Contin" 9.53 0.0500 0.1940 4.46 3.16 6.68 "-68.68" NO
"BRANCH 42163 42170 1" "Contin" 10.14 -0.0131 -0.0651 -1.39 -0.92 7.29 "-15.58" NO
</SUBDATA>
```

ATCExtraMonitor

This subdata section contains a list of the ATCExtraMonitors for this scenario. Each line contains three fields relating one of the ATCExtraMonitors. The first field describes the ATCExtraMonitor which this subdata corresponds to. The second and third variables are the initial value and sensitivity

for this extra monitor for the scenario. An option fourth field may be included if we are using one of the iterated ATC solution options. This field must be the String "ATCFlowValue".

Monitor Description: Contains a string describing which monitor this belongs to.

Initial Value: Contains the value for this extra monitor at the last linear iteration.

Sensitivity: Contains the sensitivity of this monitor.

ATCFlowValue: A string which signifies that a block will follow which stores a list of flow values for specified transfer levels. Each line of the block consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW). The block is terminated when a line of text that starts with 'END' is encountered.

Example:

```
<SUBDATA ATCExtraMonitor>  
"Interface<KEY1>Left-Right</KEY1>" 40.0735 0.633295  
"Branch<KEY1>2</KEY1><KEY2>5</KEY2><KEY3>1</KEY3>" 78.7410 0.266589  
</SUBDATA>
```

Data AuxFileExportFormatData

DataBlockDescription

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

ObjectType: Name of the object to include in the auxiliary file.

[FieldList]: List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

[SubdataList]: List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

"Filter": Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

Example:

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
Area [AreaName, AreaNum] [] "SELECTED"
Gen [BusNum, BusName, GenID] [BidCurve, ReactiveCapability] ""
</SUBDATA>
```

DataBlockDescription

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

ObjectType: Name of the object to include in the auxiliary file.

[FieldList]: List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

[SubdataList]: List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

"Filter": Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

Example:

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
DisplayArea [AreaName, AreaNum, SOAuxiliaryID] [] ""
DisplayTransmissionLine [BusNum, BusNum:1, LineCircuit, SOAuxiliaryID] [Line] "Nominal
Voltage > 138 kV"
</SUBDATA>
```

Data BGCalculatedField

Condition

Calculated Fields allow you to define a calculation over most network and aggregation objects along with a few other types of objects. The calculation can then be used to show an aggregation calculation on objects that link to these calculation objects in some manner. Part of the definition is a filter which specifies which objects to operate over. This subdata section is identical to the Condition subdata section of the Filter object type.

Data Bus

MWMarginalCostValues**MvarMarginalCostValues****LPOPFMarginalControls**

These three sections contain specific values computed for an OPF solution. In MWMarginalCostValues or MvarMarginalCostValues these specific values are the MW or Mvar marginal prices for each constraint. In LPOPFMarginalControls the values are the sensitivities of the controls with respect to the cost of each bus.

Example:

```
<SUBDATA MWMarginalCostValues>  
//Value  
16.53  
0.00  
21.80  
</SUBDATA>
```

Data BusViewFormOptions

BusViewBusField

BusViewFarBusField

BusViewGenField

BusViewLineField

BusViewLoadField

BusViewShuntField

The values represent specific fields on the custom defined bus view onelines. Each line contains two values:

Location : The various locations on the customized bus view contain slots for fields. This is the slot number.

FieldDescription : This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

Name of Field: The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

Total Digit: Number of total digits for a numeric field.

Decimal Points: Number of decimal points for a numeric field.

Color: This is the color of the field. It is not presently used.

Increment Value: This is the "delta per mouse" click for the field.

Example:

```
<SUBDATA BusViewLineField>
0 "MW Flow@6@1@0@0"
1 "MVar Flow@6@1@0@0"
2 "MVA Flow@6@1@0@0"
3 "BusAngle:1@6@2@0@0"
</SUBDATA>
```

Data ColorMap

ColorPoint

A colorpoint is simply described by a real number (between 0 and 100) indicating the percentage breakpoint, an integer describing the color, and a field indicating if the color should be used or the contour should be transparent. These three values are written on a single line of text. Each line contains three values:

cmvalue: Real number between 0 and 100 (minimum to maximum value).

cmcolor: Integer between 0 and 16,777,216. Value is determined by taking the red, green and blue components of the color and assigning them a value between 0 and 255. The color is then equal to $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$.

cmalpha: Integer between 0 and 255, where only 0 and 255 are valid values. A value of 0 indicates that the color point is transparent, while a value of 255 indicates that the color point is opaque. If the alpha channel is omitted, a default value of 255 (opaque) will be assigned.

Example:

```
<SUBDATA ColorPoint>
// Value Color Alpha
100.0000 127 255
62.5000 65535 255
50.0000 8388479 0
12.5000 16711680 0
0.0000 8323072 255
</SUBDATA>
```

Data Contingency

CTGElementAppend

The format for this data type is the same as described below for CTGElement, with the exception that the elements listed in this data section will be added to the existing list of contingency elements, instead of replace the existing elements.

CTGElement

A contingency element is described by up to four entries. All entries must be on a single line of text.

"Action" "ModelCriteria" Status //comment

Action: String describing the action associated with this element. See below for actions available.

ModelCriteria: This is the name of a ModelFilter or ModelCondition under which this action should be performed. This entry is optional. If it is not specified, then a blank (or no criteria) is assumed. If you want to enter a Status, then use must specify "" as the ModelCriteria.

Status: Three options

CHECK: perform action if ModelCriteria is true.

ALWAYS: perform action regardless of ModelCriteria.

NEVER: do not perform action.

This entry is optional. If it is not specified, then CHECK is assumed.

Comment: All text to the right of the comment symbol (//) will be saved with the CTGElement as a comment.

Possible Actions:

Calling of a name ContingencyBlock

CONTINGENCYBLOCK | name

Calls a ContingencyBlock and executes each of the actions in that block.

Transmission Line or Transformer outage or insertion

BRANCH | bus1# bus2# ckt | OPEN

| CLOSE

| OPENCBS

| CLOSECBS

Takes branch out of service, or puts it in service. For more description of the OpenCBS action type see the [Open Breakers topic](#). For more information on how the CloseCBS action works see the [Close Breakers Overview topic](#). Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

Series Capacitor Bypass or In Service

SERIESCAP | bus1# bus2# ckt | BYPASS

| INSERVICE

Bypasses a series capacitor, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

DC Transmission Line outage or insertion

```
DCLine | bus1# bus2# ckt | OPEN  
| CLOSE  
| OPENCBS  
| CLOSECBS
```

Takes DC Line out of service, or puts it in service. For more description of the OpenCBS action type see the [Open Breakers topic](#). For more information on how the CloseCBS action works see the [Close Breakers Overview topic](#). Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

DC Line set a specific value

```
DCLINE | bus1# bus2# ckt | SET_P_TO | value | MW  
| | CHANGE_P_BY | | PERCENT  
| | SET_I_TO | | AMPS  
| | CHANGE_I_BY
```

Use to set the DC line setpoint to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

Phase Shifter set a specific value

```
PHASESHIFTER | bus1# bus2# ckt | SET_P_TO | value | MW  
| | CHANGE_P_BY | | PERCENT
```

Use to set the phase shift angle to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

3-Winding Transformer outage or insertion

```
3WXFORMER | bus1# bus2# bus3# ckt | OPEN  
| CLOSE  
| OPENCBS  
| CLOSECBS
```

Takes all three windings of a 3-winding transformer out of service, or puts them in service. For more description of the OpenCBS action type see the [Open Breakers topic](#). For more information on how the CloseCBS action works see the [Close Breakers](#)

[Overview topic](#). Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# bus3# ckt] may be replaced by the label of the three winding transformer.

Interface outage or insertion

INTERFACE | name | OPEN
| CLOSE
| OPENCBS
| CLOSECBS

Takes all monitored branches in the interface out of service, or puts them all in service. For more description of the OpenCBS action type see the [Open Breakers topic](#). For more information on how the CloseCBS action works see the [Close Breakers Overview topic](#).

Note: the [name] may be replaced by the label of the interface.

Bus outage causes all lines connected to the bus to be outage

BUS | bus# | OPEN
| OPENCBS

Takes all branches connected to the bus out of service. Also outages all generation, load, or shunts attached to the bus. For more description of the OpenCBS action type see the [Open Breakers topic](#). Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

Generator, Load, or Switched Shunt outage or insertion

GEN | bus# id | OPEN
LOAD | bus# id | CLOSE
SHUNT | bus# id | OPENCBS
| | CLOSECBS
INJECTIONGROUP | name

Takes a generator, load, or shunt out of service, or puts it in service. If specifying an injection group, the status of all devices in the injection group will be changed. For more description of the OpenCBS action type see the [Open Breakers topic](#). For more information on how the CloseCBS action works see the [Close Breakers Overview topic](#).

Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the sequence [bus1# ckt] or [name] may be replaced by the label of the device.

Generator, Load or Switched Shunt movement to another bus

GEN | bus1# | MOVE_PQ_TO | bus2# | value | MW
LOAD | bus1# id | MOVE_P_TO | | MVAR
SHUNT | | MOVE_Q_TO | PERCENT

Use to move generation, load or shunt at a bus1 over to bus2. This can be used on a bus or specific device basis in specifying what to move. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also

be replaced by a string enclosed in single quotes which represents the label of the bus. When identifying specific devices, the device label can replace the bus number and device id.

Generator, Load or Switched Shunt set or change a specific value

GEN | bus# | SET_P_TO | value | MW
LOAD | bus# id | SET_Q_TO | | MVAR
SHUNT | | SET_PQ_TO | PERCENT
| SET_VOLT_TO |
| CHANGE_P_BY |
| CHANGE_Q_BY |
| CHANGE_PQ_BY |
| CHANGE_VOLT_BY |

Use to set the generation, load, or shunt at a bus to a particular value. This can also be used on a specific device. This can also be used to change by a specified amount. The voltage setpoints only apply to SHUNTS and GENs. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. When identifying specific devices, the device label can replace the bus number and device id.

Injection Group set a specific value

INJECTIONGROUP | name | SET_P_TO | value | MW
| PERCENT

Use to set the MW generation/load in an injection group to a particular value.

Injection Group change specific value

INJECTIONGROUP | name | CHANGE_P_TO | value | MW
| PERCENT
| MWMERITORDER
| PERCENTMERITORDER
| MWMERITORDEROPEN
| PERCENTMERITORDEROPEN

Use to change the MW generation/load in an injection group by a particular value.

MW and PERCENT : will change each point in the injection group by a value in proportion to the participation factors of the group.

MWMERITORDER and PERCENTMERITORDER : will modify only generators points in the injection group. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum MW limit before moving on to the next generator. This process continues until the desired injection is met. Generators will not be opened in the process so all generators will continue to provide Mvar support.

MWMERITORDEROPEN and PERCENTMERITORDEROPEN : will modify only generator points in the injection group. When chosen and the change in the MW output is *negative*, then the merit order dispatch will be modified. In this case, the generator in the injection group with the highest participation factor will have its status changed to Open, followed by the second generator and so on. This will continue until the amount of MW opened is

as close to the desired amount without exceeding the desired amount of drop. If a generator would cause the desired gen drop amount to be exceeded, that generator is skipped and the next generator in merit order is processed. If the change requested is *positive*, then generators will be increased toward their maximum MW output in the same manner as though this option were not chosen.

Note: the [name] may be replaced by the label of the injection group.

Execute a Power Flow Solution

SOLVEPOWERFLOW

Include this action to cause the solution of the contingency to be split into pieces.

Actions that are listed before each SOLVEPOWERFLOW call will be performed as a group.

Make-Up Power Compensation.

Only valid immediately following a SET, CHANGE, OPEN or CLOSE action on a Generator, Shunt or Load. This describes how the change in MW or MVAR are picked up by buses throughout the system. The values specify participation factors

COMPENSATION

bus#1 value1

bus#2 value2

...

END

Example:

<SUBDATA CTGElement>

// just some comments

// action	Model Criteria	Status	comment
"BRANCH 40821 40869 1 OPEN"	""	ALWAYS	//Raver - Paul 500 kV
"GEN 45041 1 OPEN"	""	ALWAYS	//Trip Unit #2
"BRANCH 42702 42727 1 OPEN"	"Line X Limited"	CHECK	//Open Fern Hill
"GEN 40221 1 OPEN"	"Interface L1"	CHECK	//Drop ~600 MW
"GEN 40227 1 OPEN"	"Interface L2"	CHECK	//Drop ~1200 MW
"GEN 40221 1 OPEN"	"Interface L3"	CHECK	//Drop ~600 MW
"GEN 40227 1 OPEN"	"Interface L3+"	CHECK	//Drop ~1200 MW

</SUBDATA>

Note: ContingencyElement object types can also be created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ContingencyElement belongs.

LimitViol

A LimitViol is used to describe the results of a contingency analysis run. Each Limit Violation lists eight values:

ViolType ViolElement Limit ViolValue PTDF PTDF InitialValue Reason

ViolType: One of six values describing the type of violation

BAMP: branch amp limit violation

BMVA: branch MVA limit violation

VLOW: bus low voltage limit violation

VHIGH: bus high voltage limit violation

INTER: interface MW limit violation

CUSTOM: Custom Monitor value

ViolElement: This field depends on the ViolType.

for VLOW, VHIGH: "bus1#" or "busname_buskv" or "buslabel"

for INTER: "interfacename" or "interfacelabel"

for BAMP, BMVA: "bus1# bus2# ckt violationbus# MWFlowDirection"

violationbus# is the bus number for the end of the branch which is violated

MWFlowDirection is the direction of the MW flow on the line. Potential values are "FROMTO" or "TOFROM"

Note: each bus# may be replaced with the name underscore nominal kV string enclosed in single quotes, or bus# values may be replaced by a string enclosed in single quotes representing the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

for CUSTOM: "custommonitorname deviceidentifier" where the deviceidentifier will use the key fields or label as specified by the option selected when saving

Limit: This is the numerical limit which was violated.

ViolValue: This is the numerical value of the violation.

PTDF: This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during in the base case with respect to a transfer direction This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

OTDF: This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during this contingency with respect to a transfer direction. This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

InitialValue: This stores a number. This stores the base case value for the element which is being violated. This is used to compare against when looking at change violations.

Reason: This will say whether this was a pure violation or is being reported as a violation because the change from the base case is higher than a specified threshold.

LIMIT - means this is a violation of a line/interface/bus limit or simply a Custom Monitor

CHANGE - means this is being reported as a limit because the change from the initial value is higher than allowed

Example:

<SUBDATA LimitViol>

```
BAMP "1 3 1 1 FROMTO" 271.94031 398.48096 10.0 15.01 // Note OTDF/PTDF
INTER "Right-Top" 45.00000 85.84451 // Note OTDF/PTDF not included
</SUBDATA>
```

Note: ViolationCTG object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ViolationCTG belongs.

Sim_Solution_Options

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim_Solution_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

1. Contingency Record Options (stored with the particular contingency)
2. Contingency Tool Options (stored with CTG_Options)
3. The global solution options

WhatOccurredDuringContingency

Each line of this subdata section is part of a text description of what actually ended up being implemented for this contingency. This will list which actions were executed and which actions ended up being skipped because of their model criteria. Each line of the subdata section must be enclosed in quotes.

Example:

```
<SUBDATA WhatOccurredDuringContingency>
"Applied: "
" OPEN Branch Two (2) TO Five (5) CKT 1 | | CHECK | | ELEMENT"
</SUBDATA>
```

ContingencyMonitoringException

Each line of this subdata section contains a string identifying a specially handled monitored element for this contingency followed by a string indicating how this monitored element should be handled with this contingency. The elements can be identified by their primary or secondary key fields or by label. The element descriptions should be enclosed in quotes because they contain spaces.

Example:

```
<SUBDATA ContingencyMonitoringException>
"Branch '2' '3' '1'" "Exclude"
"Branch 'Three_138.00' 'Four_138.00'" "Include"
"Branch 'Line_2_5'" "Default"
</SUBDATA>
```

Data CTG Options

Sim_Solution_Options

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim_Solution_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

- Contingency Record Options (stored with the particular contingency)
- Contingency Tool Options (stored with CTG_Options)
- The global solution options

Data CTGElementBlock

CTGElement

This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

CTGElementAppend

When a subdata section is defined as **CTGElementAppend** rather than **CTGElement**, the actions of this subdata section will be appended to the contingency actions, instead of replacing them. This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

Note: CTGElementBlockElement object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency block to which the CTGElementBlockElement belongs.

Data CustomCaseInfo

ColumnInfo

Each line of this SUBDATA section can be used for specifying the column width of particular columns of the respective Custom Case Information Sheet. The line contains two values – the column and then a column width. This is shown in the following example.

Example:

```
<SUBDATA ColumnInfo>  
"SheetCol" 133  
"SheetCol:1" 150  
"SheetCol:2" 50  
</SUBDATA>
```

Data CustomColors

CustomColors

These describe the customized colors used in Simulator, which are specified by the user. A custom color is an integer describing a color. Each custom color is written on a single line of text and is an integer between 0 and 16,777,216. The value is determined by taking the red, green, and blue components of the color and assigning them a value between 0 and 255. The color is then equal to $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$. Each line contains only one integer that corresponds to the color specified.

Example:

```
<SUBDATA CustomColors>  
9823301  
8613240  
</SUBDATA>
```

Data DataGrid

ColumnInfo

Contains a description of the columns which are shown in the respective data grid. Each line of text contains four fields: **VariableName, ColumnWidth, TotalDigits, DecimalPoints**

VariableName: Contains the variable which is shown in this column.

ColumnWidth: The column width.

TotalDigits: The total digits displayed for numerical values.

DecimalPoints: The decimal points shown for numerical values.

Example:

```
DATA (DataGrid, [DataGridName])
```

```
{
```

```
  BUS
```

```
  <SUBDATA COLUMNINFO>
```

```
  BusNomVolt 100 8 2
```

```
  AreaNum 50 8 2
```

```
  ZoneNum 50 8 2
```

```
  </SUBDATA>
```

```
  BRANCHRUN
```

```
  <subdata COLUMNINFO >
```

```
  BusNomVolt: 0 100 8 2
```

```
  BusNomVolt: 1 100 8 2
```

```
  LineMW: 0 100 9 3
```

```
  </SUBDATA>
```

```
}
```

Data DynamicFormatting

DynamicFormattingContextObject

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

DisplayObjectType: The object type of the display object. These are generally the same as the values seen in the subdata section *SelectByCriteriaSetType* of *SelectByCriteriaSet* object types. The only exception is the string *CaseInfo*, which is used for formatting applying to the case information displays.

(WhichFields) : For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

(ListOfFields) : If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

Example:

```
<SUBDATA DynamicFormattingContextObject>
// Note: CaseInfo applies to case information displays
CaseInfo "SPECIFIED" BusName
DisplayAreaField "ALL"
DisplayBus
DisplayBusField "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker
DisplaySubstation
DisplaySubstationField "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine
DisplayTransmissionLineField "ALL"
</SUBDATA>
```

LineThicknessLookupMap

LineColorLookupMap

FillColorLookupMap

FontColorLookupMap

FontSizeLookupMap

BlinkColorLookupMap

XoutColorLookupMap

FlowColorLookupMap

SecondaryFlowColorLookupMap

The values of the lookup table for the characteristics that can be modified by the dynamic formatting tool. The first line contains the two following fields:

fieldname: It is the field that the lookup table is going to look for.

usediscrete: Set to YES or NO. If set to YES, the characteristic values will be discrete, meaning that the characteristic value will correspond exactly to the one specified in the table. If set to NO, the characteristic values will be continuous, which means the characteristic value will be an interpolation of the high and low closest values specified in the table.

The following lines contain two fields:

fieldvalue: The value for the field.

characteristicvalue: The corresponding characteristic value for such field value.

Example:

```
<SUBDATA LineColorLookupMap>
// FieldName UseDiscrete
BusPUVOLT YES
// FieldValue Color
1.02 16711808
1.05 8454143
1.1 16744703
</SUBDATA>
```

Data Filter

Condition

Conditions store the conditions of the filter. Each condition is described by one line of text which can contain up to five fields.

Variablename condition "value" ("othervalue") (fieldopt)

Variablename: It is one of the fields for the object_type specified. It may optional be followed by a colon and a non-negative integer. If not specified, 0 is assumed.

Example: on a LINE, 0 = from bus, 1 = to bus

Thus: LineMW: 0 = the MW flow leaving the from bus

LineMW: 1 = the MW flow leaving the to bus

Condition: Possible Values Alternate1 Alternate2 Requirements

between >< requires other value

notbetween ~>< requires other value

equal ==

notequal <> ~=

greaterthan >

lessthan <

greaterthanorequal >=

lessthanorequal <=

contains

notcontains

startswith

notstartswith

inrange

notinrange

meets

notmeets

isblank

notisblank

value: The value to compare to.

For fields associated with strings, this must be a string

For fields associated with real numbers, this must be a number.

For fields associated with integers, this is normally an integer, except when the Condition is "inrange" or "notinrange". In this case, value is a comma/dash separated number string.

(othervalue): If required, the other value to compare to.

(FieldOpt): Optional integer value with following meanings.

0 - strings are case insensitive, use number fields directly

(0 is the default value if not otherwise specified)
1 - strings are case sensitive, take ABS of field values

Example:

```
DATA (FILTER, [objecttype, filtername, filterlogic, filterpre])
{
BUS "a bus filter" "AND" "no"
<SUBDATA CONDITION>
BusNomVolt > 100
AreaNum inrange "1-5 , 7 , 90-95"
ZoneNum between
</SUBDATA>

BRANCH "a branch filter" "OR" "no"
<subdata CONDITION>
BusNomVolt: 0 > 100 // Note location 0 means from bus
BusNomVolt: 1 > 100 // Note location 1 means to bus
LineMW: 0 > 100 1 // Note, final field 1 denotes absolute value
</SUBDATA>
}
```

Data Gen

BidCurve

BidCurve subdata is used to define a piece-wise linear cost curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost).

Example:

```
<SUBDATA BidCurve>
// MW Price[$/MWhr]
100.00 10.6
200.00 12.4
400.00 15.7
500.00 16.0
</SUBDATA>
```

ReactiveCapability

Reactive Capability subdata is used to the reactive capability curve of the generator. Each line of text consists of three real numbers: a MW output, and then the respective Minimum MVAR and Maximum MVAR output.

Example:

```
<SUBDATA ReactiveCapability>
// MW MinMVAR MaxMVAR
100.00 -60.00 60.00
200.00 -50.00 50.00
400.00 -30.00 20.00
500.00 -5.00 2.00
</SUBDATA>
```

Note: ReactiveCapability object types can also be directly created inside their own DATA section as well. Two of the key fields of the object are then the bus number and generator ID of the generator to which the ReactiveCapability point belongs.

TotalAreaValueMap

This subdata section is used to define the lookup table for determining the total area size of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Total Area attribute.

TotalArea: The total area size of the object.

Example:

```
<SUBDATA TotalAreaValueMap>
// FieldValue TotalArea
1.000 0
4.000 23
7.000 46
</SUBDATA>
```

RotationRateValueMap

This subdata section is used to define the lookup table for determining the rotation rate of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Rotation Rate attribute.

RotationRate: The rotation rate of the object. Entered in Hz.

Example:

```
<SUBDATA RotationRateValueMap>
// FieldValue RotationRate
1.000 0.00
4.000 0.10
7.000 0.20
</SUBDATA>
```

RotationAngleValueMap

This subdata section is used to define the lookup table for determining the rotation angle of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Rotation Angle attribute.

RotationAngle: The rotation angle of the object. Entered in degrees.

Example:

```
<SUBDATA RotationAngleValueMap>
```

```
// FieldValue RotationAngle
1.000 -90.0
4.000 0.0
7.000 90.0
</SUBDATA>
```

LineThicknessValueMap

This subdata section is used to define the lookup table for determining the thickness of the border line around geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

FieldValue: Value of the field selected for the Line Thickness attribute.

LineThickness: The line thickness of the border line around the object. This should be an integer value.

Example:

```
<SUBDATA LineThicknessValueMap>
// FieldValue LineThickness
1.000 1
4.000 2
7.000 3
</SUBDATA>
```

Data GlobalContingencyActions

CTGElementAppend

This format is the same as for the Contingency objecttype

CTGElement

This format is the same as for the Contingency objecttype

Note: GlobalContingencyActionsElement object types can also be directly created inside their own DATA section as well.

HintObject

Stores the values for the custom hints. Each line has one value:

FieldDescription: This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

Name of Field: The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

Total Digit: Number of total digits for a numeric field.

Decimal Points: Number of decimal points for a numeric field.

Include Suffix: Set to 0 for not including the suffix, and set to 1 to include it.

Field Prefix: The prefix text.

Example:

```
<SUBDATA HintObject>
"BusPUVolt@4@1@1@PU Volt ="
"BusAngle@4@1@1@Angle ="
</SUBDATA>
```

Data InjectionGroup

PartPoint

A participation point is used to describe the contents of an injection group. Each participation point lists six values:

PointType PointBusNum PointID PointParFac ParFacCalcType ParFacNotDynamic

PointType: One of two values describing the type of violation:

GEN: a generator

LOAD: a load

SHUNT: a switched shunt

INJECTIONGROUP: another injection group

PointBusNum: The bus number of the partpoint if the type is GEN, LOAD, or SHUNT. Value will be blank for an injection group type. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in double quotes that represents the label of the bus or a string representing the label of the generator, load, or switched shunt.

PointID: For a GEN, LOAD, or SHUNT type, this is the id for the partpoint. For an INJECTIONGROUP type, this is the name or label of the injection group.

PointParFac: The participation factor for the point.

ParFacCalcType: How the participation point is calculated. There are several options depending on the PointType.

Generators: SPECIFIED, MAX GEN INC, MAX GEN DEC, or MAX GEN MW

Loads: SPECIFIED or LOAD MW

Shunts: SPECIFIED, MAX SHUNT INC, MAX SHUNT DEC, or MAX SHUNT MVAR

Injection Groups: SPECIFIED

ParFacNotDynamic: Should the participation factor be recalculated dynamically as the system changes.

Example:

```
<SUBDATA PartPoint
"GEN" 1 "1" 1.00 "SPECIFIED" "NO"
"GEN" 2 "1" 5.00 "SPECIFIED" "NO"
"GEN" 4 "1" 104.96 "MAX GEN INC" "NO"
"GEN" 6 "1" 50.32 "MAX GEN DEC" "YES"
"GEN" 7 "1" 600.00 "MAX GEN MW" "NO"
"LOAD" 2 "1" 5.00 "SPECIFIED" "NO"
"LOAD" 5 "1" 130.00 "LOAD MW" "NO"
"LOAD" 6 "1" 200.00 "LOAD MW" "YES"
</SUBDATA>
```

Note: PartPoint object types can also be directly created inside their own DATA section as well. One of the key fields of the PartPoint object is then the name of the injection group to which the participation point belongs.

Data Interface

InterfaceElement

An interface's subdata contains a list of the elements in the interface. Each line contains a text description of the interface element. Note that this text description must be encompassed by quotation marks. There are five kinds of elements allowed in an interface. Please note that the direction specified in the monitoring elements is important.

"BRANCH num1 num2 ckt" Monitor the MW flow on the branch starting from bus num1 going to bus num2 on the branch this branch. (order of bus numbers defines the direction)

"AREA num1 num2" Monitor the sum of the AC tie line MW flows from area num1 to area num2.

"ZONE num1 num2" Monitor the sum of the AC tie line MW flows from zone num1 to zone num2.

"BRANCHOPEN num1 num2 ckt" When monitoring the elements in this interface, monitor them under the contingency of opening this branch.

"BRANCHCLOSE num1 num2 ckt" When monitoring the elements in this interface, monitor them under the contingency of closing this branch.

"DCLINE num1 num2 ckt" Monitor the flow on a DC line.

"INJECTIONGROUP 'name'" Monitor the net injection from an injection group (generation contributes a positive injection, loads negative).

"GEN num1 id" Monitor the net injection from a generator (output is positive injection).

"LOAD num1 id" Monitor the net injection from a load (output is negative injection).

"MSLINE num1 num2 ckt" Monitor the MW flow on the multi-section line starting from bus num1 going to bus num2 with circuit ckt.

"INTERFACE 'name'" Monitor the MW flow on the interface given by name.

Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. Labels may also be used as follows:

- bus# values for all element types may be replaced by a string enclosed in single quotes where the string is the label of the bus
- for GEN or LOAD elements, the section *num1 id* may be replaced by the device's label
- for MSLINE, DCLINE, or BRANCH elements, the *num1 num2 ckt* section may be replaced by the device's label

For the interface element type "BRANCH num1 num2 ckt" and "DCLINE num1 num2 ckt", an optional field can also be written specifying whether the flow should be measured at the far end. This field is either YES or NO.

Example:

```
<SUBDATA InterfaceElement
```

```
"BRANCH 8 9 1" NO // monitor the flow from bus 8 to bus 9 on this branch
```

```
"BRANCH 12 33 1" YES // monitor the flow from bus 12 to bus 33 on branch
```

```
// measurefarend is set to true, therefore, we are
// monitoring the MW flow that arrives at bus 33

"AREA 2 1" // monitor tie line flow from area 2 to area 1
"ZONE 66 53" // monitor tie lines flows from zone 66 to zone 53
"BRANCHOPEN 5 6 1" // doe monitoring after branch opens
"BRANCHCLOSE 7 10 1" // doe monitoring after branch closes
</SUBDATA>
```

Note: InterfaceElement object types can also be directly created inside their own DATA section as well. One of the key fields of the InterfaceElement object is then the name of the interface to which the interface element belongs.

Data KMExportFormat

DataBlockDescription

This subdata section is used to describe the objects and fields that should be saved to a KML file. Same format as for the AUXFileExportFormatData subdata section.

Data LimitSet

LimitCost

LimitCost records describe the piece-wise unenforceable constraint cost records for use by unenforceable line/interface limits in the OPF or SCOPF. Each row contains two values

PercentLimit: Percent of the transmission line limit.

Cost: Cost used at this line loading percentage value.

Example:

```
<SUBDATA LimitCost>  
//Percent Cost [$/MWhr]  
100.00 50.00  
105.00 100.00  
110.00 500.00  
</SUBDATA>
```

Data Load

BidCurve

BidCurve subdata is used to define a piece-wise linear benefit curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost). These costs must be increasing for loads.

Example:

```
<SUBDATA BidCurve>  
// MW Price[$/MWhr]  
100.00 16.0  
200.00 15.7  
400.00 12.4  
500.00 10.6  
</SUBDATA>
```

Data LPVariable

LPVariableCostSegment

Stores the cost segments for the LP variables. Each line contains four values:

Cost (Up): Cost associated with increasing the LP variable.

Minimum value: Minimum limit of the LP variable.

Maximum value: Maximum limit of the LP variable.

Artificial: Whether the cost segment is artificial or not.

Data ModelCondition

Condition

ModelConditions are the combination of an object and a Filter. They are used to return when the particular object meets the filter specified. As a results, the subdata section here is identical to the Condition subdata section of a [Filter](#). See the description there.

Data ModelExpression

LookupTable

LookupTables are used inside Model Expressions sometimes. These lookup table represent either one or two dimensional tables. If the first string in the SUBDATA section is "x1x2", this signals that it is a two dimensional lookup table. From that point on it will read the first row as "x2" lookup points, and the first column in the remainder of the rows as the x1 lookup values.

Example:

```
DATA (MODELEXPRESSION, [CustomExpression,ObjectType,CustomExpressionStyle,  
CustomExpressionString,WhoAmI,VariableName,WhoAmI:1,VariableName:1], AUXDEF)
```

```
{  
// The following demonstrated a one dimensional lookup table  
22.0000, "oneD", "Lookup", "",  
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",  
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",  
"", ""  
<SUBDATA LookupTable>  
// because it does not start with the string x1x2 this will  
// represent a one dimensional lookup table  
x1 value  
0.000000 1.000000  
11.000000 22.000000  
111.000000 222.000000  
</SUBDATA>
```

```
0.0000, "twod", "Lookup", "",  
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",  
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",  
"Gen<KEY1>6</KEY1><KEY2>1</KEY2>",  
"Gen<KEY1>6</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>"  
<SUBDATA LookupTable>  
// because this starts with x1x2 this represent a two dimensional  
// lookup table. The first column represents lookup values for x1.  
// The first row represents lookup values for x2  
x1x2 0.100000 0.300000 // these are lookup heading for x2  
0.000000 1.000000 3.000000  
11.000000 22.000000 33.000000  
111.000000 222.000000 333.000000  
</SUBDATA>  
}
```

Data ModelFilter

ModelCondition

A Model Filter's subdata contains a list of each ModelCondition in the filter. Because a list of Model Conditions is stored within Simulator, this subdata section only requires the name of each ModelCondition on each line and whether or not the condition is using the NOT operator as part of the Model Filter.

Example:

```
<SUBDATA ModelCondition>
// ModelConditionName NotCondition
"Name of First Model Condition" "NO"
"Name of Second Model Condition" "NO"
"Name of Third Model Condition" "NO"
</SUBDATA>
```

Data MTDCRecord

MTDCBus

For this SUBDTA section, each DC Bus is described on a single line of text with exactly 8 fields specified.

DCBusNum: The number of the DC Bus. Note the DC bus numbers are independent of the AC bus numbers.

DCBusName: The name of the DC bus enclosed in quotes.

ACTerminalBus: The AC terminal to which this DC bus is connected (via a MTDCConverter). If the DC bus is not connected to any AC buses, then specify as zero. You may also specify this as a string enclosed in double quotes with the bus name followed by an underscore character, following by the nominal voltage of the bus.

DCResistanceToground: The resistance of the DC bus to ground. Not used by Simulator.

DCBusVoltage: The DC bus voltage in kV.

DCArea: The area that this DC bus belongs to.

DCZone: The zone that this DC bus belongs to.

DCOwner: The owner that this DC bus belongs to.

Note: MTDCBus object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCBus belongs.

MTDCConverter

For this SUBDTA section, each AC/DC Converter is described by exactly 24 field which may be spread across several lines of text. Simulator will keep reading lines of text until it finds 24 fields. All text to the right of the 24th field (on the same line of text) will be ignored. The 24 fields are listed in the following order:

BusNum: AC terminal bus number.

MTDCNBridges: Number of bridges for the converter.

MTDCConvEBas: Converter AC base voltage.

MTDCConvAngMxMn: Converter firing angle.

MTDCConvAngMxMn:1: Converter firing angle max.

MTDCConvAngMxMn:2: Converter firing angle min.

MTDCConvComm: Converter commutating resistance.

MTDCConvComm:1: Converter commutating reactance.

MTDCConvXFRat: Converter transformer ratio.

MTDCFixedACTap: Fixed AC tap.

MTDCConvTapVals: Converter tap.

MTDCConvTapVals:1: Converter tap max.

MTDCConvTapVals:2: Converter tap min.

MTDCConvTapVals:3: Converter tap step size.

MTDCConvSetVL: Converter setpoint value (current or power).

MTDCConvDCPF: Converter DC participation factor.

MTDCConvMarg: Converter margin (power or current).

MTDCCConvType: Converter type.

MTDCMaxConvCurrent: Converter Current Rating.

MTDCCConvStatus: Converter Status.

MTDCCConvSchedVolt: Converter scheduled DC voltage.

MTDCCConvIDC: Converter DC current.

MTDCCConvPQ: Converter real power.

MTDCCConvPQ:1: Converter reactive power.

Note: MTDCConverter object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCConverter belongs.

MTDCTransmissionLine

For this SUBDATA section, each DC Transmission Line is described on a single line of text with exactly 5 fields specified:

DCFromBusNum: From DC Bus Number.

DCToBusNum: To DC Bus Number.

CKTID: The DC Circuit ID.

Resistance: Resistance of the DC Line in Ohms.

Inductance: Inductance of the DC Line in mHenries (Not used by Simulator).

Example:

```
DATA (RECORD, [Num,Mode,ControlBus])
{
//-----
// The first Multi-Terminal DC Transmission Line Record
//-----
1 "Current" "SYLMAR3 (26098)"
<SUBDATA Bus>
//-----
// DC Bus data must appear on a single line of text
// The data consists of exactly 8 values
// DC Bus Num, DC Bus Name, AC Terminal Bus, DC Resistance to ground,
// DC Bus Voltage, DC Bus Area, DC Bus Zone, DC Bus Owner
//-----
3 "CELILO3P" 0 9999.00 497.92 40 404 1
4 "SYLMAR3P" 0 9999.00 439.02 26 404 1
7 "DC7" 41311 9999.00 497.93 40 404 1
8 "DC8" 41313 9999.00 497.94 40 404 1
9 "DC9" 26097 9999.00 439.01 26 404 1
10 "DC10" 26098 9999.00 439.00 26 404 1
</SUBDATA>
<SUBDATA Converter>
```

```

//-----
// convert subdata keeps reading lines of text until it has found
// values specified for 24 fields. This can span any number of lines
// any values to the right of the 24th field found will be ignored
// The next converter will continue on the next line.
//-----
41311 2 525.00 20.25 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3274
41313 4 232.50 15.36 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 561.8186
26097 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26098 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
//-----
// DC Transmission Segment information appears on a single line of
// text. It consists of exactly 5 value
// From DCBus, To DCBus, Circuit ID, Line Resistance, Line Inductance
//-----
3 4 "1" 19.0000 1300.0000
7 3 "1" 0.0100 0.0000
8 3 "1" 0.0100 0.0000
9 4 "1" 0.0100 0.0000
10 4 "1" 0.0100 0.0000
</SUBDATA>

//-----
// A second Multi-Terminal DC Transmission Line Record
//-----
2 "Current" "SYLMAR4 (26100)"

```

```

<SUBDATA Bus>
5 "CELILO4P" 0 9999.00 497.92 40 404 1
6 "SYLMAR4P" 0 9999.00 439.02 26 404 1
11 "DC11" 41312 9999.00 497.93 40 404 1
12 "DC12" 41314 9999.00 497.94 40 404 1
13 "DC13" 26099 9999.00 439.01 26 404 1
14 "DC14" 26100 9999.00 439.00 26 404 1
</SUBDATA>
<SUBDATA Converter>
41312 2 525.00 20.26 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3969
41314 4 232.50 15.45 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 562.9448
26099 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26100 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
5 6 "1" 19.0000 1300.0000
11 5 "1" 0.0100 0.0000
12 5 "1" 0.0100 0.0000
13 6 "1" 0.0100 0.0000
14 6 "1" 0.0100 0.0000
</SUBDATA>
}

```

Note: MTDCTransmissionLine object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCTransmissionLine belongs.

Data MultiSectionLine

Bus

A multi section line's subdata contains a list of each dummy bus, starting with the one connected to the From Bus of the MultiSectionLine and proceeding in order to the bus connected to the To Bus of the Line. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

Example:

```
//-----  
// The following describes a multi-section line that connects bus  
// 2 - 1 - 5 - 6 - 3  
//-----  
DATA (MultiSectionLine, [BusNum, BusName, BusNum: 1, BusName: 1,  
  LineCircuit, MSLineNSections, MSLineStatus] )  
{  
  2 "Two" 3 "Three" "&1" 2 "Closed"  
  <SUBDATA Bus>  
  1  
  5  
  6  
  </SUBDATA>  
}
```

Data Nomogram

InterfaceElementA

InterfaceElementB

InterfaceElementA values represent the interface elements for the first interface of the nomogram. InterfaceElementB values represent the interface elements for the second interface of the nomogram. The format of these SUBDATA sections is identical to the format of the InterfaceElement SUBDATA section of a normal Interface.

NomogramBreakPoint

This subdata section contains a list of the vertex points on the nomogram limit curve.

Example:

```
<SUBDATA NomogramBreakPoint>
// LimA LimB
-100 -20
-100 100
80 50
60 -10
</SUBDATA>
```

Data NomogramInterface

InterfaceElement

This follows the same convention as the InterfaceElement SUBDATA section described with the [Interface](#) objecttype.

Data Owner

Bus

This subdata section contains a list of the buses which are owned by this owner. Each line of text contains the bus number. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

Example:

```
<SUBDATA Bus>
1
35
65
</SUBDATA>
```

Load

This subdata section contains a list of the loads which are owned by this owner. Each line of text contains the bus number followed by the load id. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and load id, the label of the load enclosed in double quotes may be used.

Example:

```
<SUBDATA Load>
5 1 // shows ownership of the load at bus 5 with id of 1
423 1
</SUBDATA>
```

Gen

This subdata section contains a list of the generators which are owned by this owner and the fraction of ownership. Each line of text contains the bus number, followed by the gen id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and generator id, the label of the generator enclosed in double quotes may be used.

Example:

```
<SUBDATA Gen>
78 1 50 // shows 50% ownership of generator at bus 78 with id of 1
23 3 70
```


</SUBDATA>

Branch

This subdata section contains a list of the branches which are owned by this owner and the fraction of ownership. Each line of text contains the from bus number, followed by the to bus number, followed by the circuit id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus numbers, strings enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the two numbers and a circuit id, the label of the branch enclosed in double quotes may be used.

Example:

<SUBDATA Branch>

6 10 1 50 // shows 50% ownership of line from bus 6 to 10, circuit 1

</SUBDATA>

Data PieChartGaugeStyle

ColorMap

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

Percentage: This is the percentage at which the following scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie chart/gauge.

Color: A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>
//Percentage Scalar Color
85.0000 1.5000 33023
100.0000 2.0000 255
</SUBDATA>
```

Data PostPowerFlowActions

CTGElementAppend

This format is the same as for the Contingency objecttype.

CTGElement

This format is the same as for the Contingency objecttype.

Note: PostPowerFlowActionsElement object types can also be directly created inside their own DATA sections as well.

Data PWCaseInformation

PWCaseHeader

This subdata section contains the Case Description in free-formatted text. Note: as it is read back into Simulator all spaces from the start of each line are removed.

Data PWFormOptions

PieSizeColorOptions

There can actually be several PieSizeColorOptions subdata sections for each PWFormOptions object. The first line of each subdata section, the first line of text consist of exactly four values

ObjectName: The objectname of the type of object these settings apply to. Will be either be BRANCH or INTERFACE.

FieldName: The fieldname for the pie charts that these settings apply to.

UseDiscrete: Set to YES to use a discrete mapping of colors and size scalars instead of interpolating for intermediate values.

UseOtherSettings: Set to YES to default these settings to the BRANCH MVA values for BRANCH object. This allows you to apply the same settings to all pie charts.

After this first line of text, if the UseOtherSettings Value is NO, then another line of text will contain exactly three values:

ShowValue: This is the percentage at which the value should be drawn on the pie chart.

NormalSize: This is the scalar size multiplier which should be used for pie charts below the lowest percentage specified in the lookup table.

NormalColor: This is the color which should be used for pie charts below the lowest percentage specified in the lookup table.

Finally the remainder of the subdata section will contain a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values

Percentage: This is the percentage at which the follow scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie charts.

Color: A color for the pie charts.

Data PWLPOPFGTGViol

OPFControlSense**OPFBusSenseP****OPFBusSenseQ**

This stores the control sensitivities for each contingency violation during OPF/SCOPF analysis. Each line contains one value:

Sensitivity: The value of the sensitivity with respect to each control in OPFControlSense or with respect to each bus in OPFBusSenseP and OPFBusSenseQ.

Example:

```
<SUBDATA OPFControlSense>  
// Value  
1.000441679  
2.447185E-7  
-1.1109307E-6  
1.6427327E-7  
0  
</SUBDATA>
```

Data PWLPTabRow

LPBasisMatrix

This subdata section stores the basis matrix associated with the final LP OPF solution. Each line contains two values:

Variable: The basic variable.

Value: The sensitivity of the constraint to the basic variable.

Example:

```
<SUBDATA LPBasisMatrix>
// Var Value
1 1.00000
2 1.00000
5 1.00000
6 1.00000
</SUBDATA>
```

Data PWPVResultListContainer

PWPVResultObject

This subdata section contains the results of a particular PV Curve scenario (a particular contingency or the base case). The data consists of two general sections: the first three rows of text contain the "independent axis" of the PV Curve. The first row starts with the string INDNOM and is followed by a list of numbers representing the nominal shift, the second row starts with INDEXP and is followed by the export shift, and the third row starts with INDIMP and is followed by the import shift. Following these rows is a list of all the tracked quantities. Each tracked quantity row consists of three parts which are separated by the strings *?f=* and *&v=* . The first part of the string represents a description of the power system object being tracked, the second part represents the field name being tracked, and the third contains a list of all the values at the various shift levels.

Example:

```
<SUBDATA PWPVResultObject>
INDNOM 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDEXP 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDIMP 0.00 -417.23 -701.58 -890.58 -952.60 -975.35 -990.43
Bus '3'?f=BusPUVolt&v= 0.993 0.983 0.964 0.939 0.926 0.919 0.914
Bus '5'?f=BusPUVolt&v= 1.007 1.000 0.982 0.956 0.940 0.932 0.926
Gen '4' '1'?f=GenMVR&v= 19.99 245.27 523.62 831.13 986.84 1060.6 1118.7
Gen '6' '1'?f=GenMVR&v= -6.59 -120.84 -131.37 -39.53 48.35 103.8 154.5
</SUBDATA>
```

LimitViol

This subdata section contains the limit violations of a particular PV Curve scenario. This subdata section would only exist if using the option to monitor limit violations with the PV tool. Each row consists of an identifier, either VLOW or VHIGH, to indicate the type of limit violation followed by the bus identifier based on the key field identifier chosen. The bus can be identified by number, name and nominal kV combination, or label. The bus identifier is followed by the limit in use to identify a voltage violation and this is followed by the voltage at the bus.

Example:

```
<SUBDATA LimitViol>
VLOW 3 1.00000 0.99017
VLOW 5 1.00000 0.98245
</SUBDATA>
```

PVBusInadequateVoltages

This subdata section contains a list of buses that are considered to have inadequate voltages at each transfer level for a particular PV Curve scenario. This subdata section would only exist if using the option to store inadequate voltages. The data consists of two general sections: the first row starts with the string INDNOM and is followed by a list of numbers representing the nominal shift. The second and subsequent rows list the buses and inadequate voltages for any bus that has an

inadequate voltage at any transfer level. Each row starts with the bus identifier followed by the voltages at that bus at the corresponding shift levels. If a voltage is not inadequate at a particular transfer level, a blank entry will appear instead of a voltage value. The bus identifier is based on the key field identifier chosen and can be number, name and nominal kV combination, or label.

Example:

```
<SUBDATA PVBUSInadequateVoltages>
// INDNOM ShiftLevel1 ShiftLevel2 ...
// BUS Voltage1 Voltage2 ...
INDNOM 0.000 100.000 200.000 300.000 400.000 500.000
"Bus '3'" 0.99269 0.99278 0.99282 0.99280 0.99273 0.99262
"Bus '4'" "" 1.00000 "" "" ""
</SUBDATA>
```

PWPVResultObject

This subdata section contains the results of a particular QV Curve scenario. These results will exist when tracking quantities with the QV curve tool. The data consists of two general sections: the first three rows of text contain the "independent axis" of the QV Curve. The first three rows start with the strings INDNOM, INDEXP, and INDIMP and are followed by a list of numbers representing the setpoint voltage representing the V of the QV curve. Following after these rows is a list of all the tracked quantities. Each tracked quantity row consists of three parts which are separated by the strings ?f= and &v= . The first part of the string represents a description of the power system object being tracked, the second part represents the field name being tracked, and the third contains a list of all the values at the various setpoint voltage levels.

Example:

```
<SUBDATA PWPVResultObject>
INDNOM 1.100 1.093 1.083 1.073 1.063
INDEXP 1.100 1.093 1.083 1.073 1.063
INDIMP 1.100 1.093 1.083 1.073 1.063
Bus '1'?f=BusPUVolt&v=1.05000 1.05000 1.05000 1.05000 1.05000
Bus '1'?f=BusKVVolt&v=144.89999 144.89999 144.89999 144.89999 144.89999 </SUBDATA>
```

Data QVCurve

QVPoints

This subdata section contains a list of the QV Curve points calculated for the respect QVCurve. Each line consists of exactly four values:

PerUnitVoltage: The per unit voltage of the bus for a QV point.

FictitiousMvar: The amount of Mvar injection from the fictitious generator at this QV point.

ShuntDeviceMvar: The Mvar injection from any switched shunts at the bus.

TotalMvar: The total Mvar injection from switched shunts and the fictitious generator.

ReservesMvar: Total amount of Mvar reserves available at the bus.

ReservesTotalMvar: Total Mvar injection from the switched shunts, fictitious generator, and available reserves.

Example:

```
DATA (QVCURVE, [BusNum,CaseName,qv_VQ0,qv_Q0,qv_Vmax,qv_QVmax,qv_VQmin,qv_Qmin,
qv_Vmin,qv_QVmin,Qinj_Vmax,Qinj_0,Qinj_min,Qinj_Vmin])
{
5 "BASECASE" 0.880 0.000 1.100 312.490 0.480 -221.072
0.180 -86.334 191.490 -77.373 -244.075 -89.562
<SUBDATA QVPoints>
// NOTE: This bus has a constant impedance
// switched shunt value of -100 Mvar at it.
// V(PU), Q(MVR), Q_shunt(MVR), Q_tot(MVR), Q_res(MVR), Q_tot_res(MVR)
1.1000, 312.4898, -121.0000, 191.4898, 0.0000, 191.4898
0.9800, 124.6619, -95.9656, 28.6963, 0.0000, 28.6963
0.7800, -96.6202, -60.7808, -157.4010, 0.0000, -157.4010
0.5800, -206.9895, -33.5960, -240.5855, 0.0000, -240.5855
0.3800, -207.4962, -14.4113, -221.9075, 0.0000, -221.9075
0.1800, -86.3336, -3.2284, -89.5620, 0.0000, -89.5620
</SUBDATA>
}
```

Data QVCurve_Options

Sim_Solution_Options

This subdata section contains solution options that will be used when running QV Curves. See explanation under the CTG_Options object type for more information.

Data SelectByCriteriaSet

SelectByCriteriaSetType

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

DisplayObjectType: The object type of the display object.

(FilterName): A filter to apply to these types of objects. This field is optional, but must be given if either of the following fields are given.

(WhichFields): For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

(ListOfFields): If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

Example:

```
<SUBDATA SelectByCriteriaSetType>
DisplayAreaField "" "ALL"
DisplayBus ""
DisplayBusField "Name of Bus Filter" "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker ""
DisplaySubstation ""
DisplaySubstationField "" "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine ""
DisplayTransmissionLineField "" "ALL"
</SUBDATA>
```

Area

This subdata section contains a list of areas which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

Example:

```
<SUBDATA Area>
18 // NEVADA
22 // SANDIEGO
30 // PG AND E
52 // AQUILA
</SUBDATA>
```

Zone

This subdata section contains a list of zones which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

Example:

```
<SUBDATA Zone>  
680 // ID SOLUT  
682 // WY NE IN  
</SUBDATA>
```

ScreenLayer

This subdata section contains a list of screen layers which were chosen to be selected. Each line of the section consists of either the name.

Example:

```
<SUBDATA ScreenLayer>  
"Border"  
"Transmission Line Objects"  
</SUBDATA>
```

Data ShapeFileExportDescription

This object uses the same subdata sections as **SelectByCriteriaSet**. The only distinction is that only buses and lines can be exported.

Data StudyMWTransactions

ImportExportBidCurve

This subdata section contains the piecewise linear transactions cost curves for areas involved in a MW transaction. Costs are only for areas that are not on OPF control. Curves must be monotonically increasing. Each line corresponds to a point in the cost curve, and it has two values:

MW: The MW value. Use negative values for imports (purchase) and positive values for exports (sales).

Price: The price in \$/MWh.

Example:

```
<SUBDATA ImportExportBidCurve>
//MW Price[$/MWh]
-20.00 5.00
-10.00 10.00
0.00 15.00
10.00 20.00
20.00 45.00
30.00 70.00
</SUBDATA>
```

Data SuperArea

SuperAreaArea

This subdata section contains a list of areas within each super area. Each line of text contains two values, the area number followed by a participation factor for the area that can be optionally used.

Example:

```
<SUBDATA SuperAreaArea>  
1 48.9  
2 34.2  
25 11.2  
</SUBDATA>
```

Data TSSchedule

SchedPointList

This section stores the schedule time points used in Time Step Simulation. Each line contains seven values:

Date: The date of the point.

Hour: The hour of the point.

Pointtype: An integer specifying the point type.

0 – Numeric

1 – Boolean (Yes/No, Closed/Open)

2 – Text

Numeric Value: The numeric value if point type is Numeric. Otherwise it is just zero.

Boolean Value: The boolean value if point type is Boolean. Otherwise it is just false.

Text Value: The text value if point type is Text. Otherwise it is just an empty string.

Audiofilename: The audio filename associated to the point. If none, it is just an empty string.

Example:

```
<SUBDATA SchedPointList>
//Date Hour PointType NValue BValue TValue AValue
5/8/2006 0 1.00 NO
5/8/2006 6:00:00 AM 0 1.10 NO
5/8/2006 12:00:00 PM 0 1.25 NO
</SUBDATA>
```

Data UserDefinedDataGrid

ColumnInfo

This follows the same convention as the ColumnInfo SUBDATA section described with the [DataGrid](#) objecttype.

Data View

ScreenLayer

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

Example:

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

Script Section

The SCRIPT section begins with a left curly brace and ends with a right curly brace.

SCRIPT ScriptName

```
{
script_statement_1
script_statement_2
.
.
.
script_statement_n
}
```

Scripts may optionally contain a ScriptName. This enables you to call a particular SCRIPT by using the [LoadScript action](#). After the optional name, the SCRIPT section begins with a left curly brace and ends with a right curly brace. Inside of this, script statements can be given.

In general, a script statement has the following format:

Keyword(arg1, arg2, ...);

- Statement starts with a keyword.
- The keyword is followed by an argument list which is encompassed in parentheses ().
- The arguments are separated by commas.
- If a single argument is a list of things, this list is encompassed by braces []. (eg. SetData)
- Statements end with a semicolon.
- Statements may take up several lines of the text file.
- You may put more than one statement on a single text line.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language. Versions of Simulator prior to 16.0 organized certain functionality with the submode feature. While existing scripts designed to work with submodes will function as before, moving between submodes is no longer necessary.

If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to **EDIT** mode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to **RUN** mode.

Script Actions for Display Auxiliary Files

The following actions are available to you regardless of the Mode:

AutoInsertBorders; AutoInsertBuses	(LocationSource, "Bus" "Substation" "File" "filename", "x,y" "lon,lat"	MapProjection, "Simple Conic" "Mercator" "x,y" FileCoordinates); "x,y" "lon,lat"	AutoInsertBranches, YES or NO	InsertIfNotAlready Shown, YES or NO
AutoInsertLoads	(MinkV,	InsertTextFields, YES or NO	InsertEquivalentObjects); YES or NO	
AutoInsertGens	(MinkV,	InsertTextFields); YES or NO		
AutoInsertSwitched Shunts	(MinkV,	InsertTextFields); YES or NO		
AutoInsertLines	(MinkV,	InsertTextFields, YES or NO	InsertEquivalentObjects, YES or NO	InsertZBRPieCharts, YES or NO
	InsertMSLines, YES or NO	SBRImpedance,	NoStubsZBRs, YES or NO	SingleCBZRs); YES or NO
AutoInsertLineFlow Objects	(MinkV,	InsertOnlyIfNotAlreadyShown, YES or NO	LineLocation, 0, 1, or 2	Size,
	FieldDigits,	FieldDecimals,	TextPosition, YES or NO	ShowMW, YES or NO
	ShowMVar, YES or NO	ShowMVA, YES or NO	ShowUnits, YES or NO	ShowComplex); YES or NO

AutoInsertSubstations	(LocationSource, "Bus" "Substation" "File" "filename",	MapProjection, "Simple Conic" "Mercator" "x,y" FileCoordinates); "x,y" "lon,lat"	AutoInsertBranches, YES or NO	InsertIfNotAlreadyShown, YES or NO
AutoInsertLineFlowPieCharts	(MinkV,	InsertOnlyIfNotAlreadyShown, YES or NO	InsertMSLines, YES or NO	Size);
AutoInsertInterfaces	(InsertPieCharts, YES or NO	PieChartSize);		
ResetStubLocations	(ZBRImpedance,	NoStubsZBRs); YES or NO		

AutoInsertBorders;

Use this action to automatically insert borders according to the settings in the AutoInsertBordersOptions object

AutoInsertBuses(LocationSource, MapProjection, AutoInsertBranches, InsertIfNotAlreadyShown, "filename", FileCoordinates);

Use this action to automatically insert buses based on specified location data.

LocationSource : "Bus", "Substation" or "File"

MapProjection : "Simple Conic", "Mercator" or "x,y"

AutoInsertBranches : YES to insert transmission lines when finished, NO not to

InsertOnlyIfNotAlreadyShown : YES if only buses that are not already shown should be inserted, NO to insert all buses.

"filename" : (optional) path to location source file (if LocationSource is "File")

FileCoordinates : (optional) format of coordinates in file "x,y" or "lon,lat" (if LocationSource is "File")

AutoInsertLoads(MinkV, InsertTextFields);

Use this action to automatically insert loads.

MinkV : Minimum kV level to insert (default=0)

InsertTextFields : (optional) insert text fields (default=YES)

AutoInsertSwitchedShunts(MinkV, InsertTextFields);

Use this action to automatically insert switched shunts.

MinkV : Minimum kV level to insert (default=0)

InsertTextFields : (optional) insert text fields (default=YES)

AutoInsertLines(MinkV, InsertTextFields, InsertEquivObjects, InsertZBRPieCharts, InsertMSLines, ZBRImpedance, NoStubsZBRs, SingleCBZRs);

Use this action to automatically insert lines.

MinkV : (optional) minimum kV level to insert (default=0)

InsertTextFields : (optional) insert text fields (default=YES)

InsertEquivObjects : (optional) insert Equivalenced Objects (default=YES)

InsertZBRPieCharts : (optional) insert pie charts for lines with no limit and bus ties
(default=NO)

InsertMSLines : (optional) insert MultiSection Lines (default=YES)

ZBRImpedance : (optional) maximum PU impedance for bus ties (default =0.0001)

NoStubsZBRs : (optional) ignore stubs for bus ties (default=YES)

SingleCBZRs : (optional) only insert a single circuit breaker (default=YES)

AutoInsertLineFlowObjects(MinkV, InsertOnlyIfNotAlreadyShown, LineLocation, Size, FieldDigits, FieldDecimals, TextPosition, ShowMW, ShowMvar, ShowMVA, ShowUnits, ShowComplex);

Use this action to automatically insert line flow objects.

MinkV : Minimum kV level to insert (default=0)

InsertOnlyIfNotAlreadyShown: (optional) if existing line flow objects are ignored
(default=YES)

LineLocation : (optional) where to insert flow objects (default=0)

0 : middle

1 : 10%/90%

2 : after stubs

Size : (optional) size (default=5.0)

FieldDigits : (optional) total digits in field (default=6)

FieldDecimals : (optional) digits to the right of the decimal (default=2)

TextPosition : (optional) position of fields relative to flow object (default=YES)

YES : above

NO : below

ShowMW : (optional) show MW field (default=YES)

ShowMvar : (optional) show Mvar field (default=YES)

ShowMVA : (optional) show MVA field (default=YES)

ShowSuffix : (optional) show field units (default=YES)

ShowComplex : (optional) show complex form (MW+jMvar) (default=NO)

AutoInsertSubStations(LocationSource, MapProjection, AutoInsertBranches, InsertIfNotAlreadyShown, "filename", FileCoordinates);

Use this action to automatically insert substations based on specified location data.

LocationSource : "Bus", "Substation" or "File"

MapProjection : "Simple Conic", "Mercator" or "x,y"

AutoInsertBranches : YES to insert transmission lines when finished, NO not to

InsertOnlyIfNotAlreadyShown : YES if only buses that are not already shown should be inserted, NO to insert all buses.

"filename" : (optional) path to location source file (if LocationSource is "File")

FileCoordinates : (optional) format of coordinates in file "x,y" or "lon,lat" (if LocationSource is "File")

AutoInsertLineFlowPieCharts(MinkV, InsertOnlyIfNotAlreadyShown, InsertMSLines, Size);

Use this action to automatically insert line flow pie charts.

MinkV : Minimum kV level to insert (default=0)

InsertOnlyIfNotAlreadyShown: (optional) if existing line flow objects are ignored (default=YES)

InsertMSLines : (optional) insert pie charts for Multi-Section Lines (default=YES)

Size : (optional) size (default=5.0)

AutoInsertInterfaces(InsertPieCharts, PieChartSize)

Use this action to automatically insert line flow objects.

InsertPieCharts : (optional) Insert pie charts as well (default=YES)

PieChartSize : (optional) default size of interface pie charts (default=50.0)

ResetStubLocations(ZBRImpedance, NoStubsZBRs)

Use this action to reset stub locations.

ZBRImpedance : (optional) max P.U. impedance for bus ties (default=0.0001)

NoStubsZBRs : (optional) Ignore stubs for bus ties (default=YES)

General Script Commands

The following script commands defined in the general [Script](#) section are available for display auxiliary files as well:

ExitProgram

LoadScript

LoadData

SelectAll

UnSelectAll

SetData
SaveData
SaveDataWithExtra
Delete
WriteToTextFile
SetData
CreateData
DeleteFile
RenameFile
CopyFile
SetCurrentDirectory
SaveObjectFields
SendToExcel
StopAuxFile

Data Section for Display Auxiliary File

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

DATA DataName (object_type, [list_of_fields], file_type_specifier, create_if_not_found)

```
{  
data_list_1  
.  
.  
.  
data_list_n  
}
```

Immediately following the DATA keyword, you may optionally include a DataName. Currently, this is not used for anything because there are no script commands that are supported with the *display auxiliary* files.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).

Data Argument List for Display Auxiliary File

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" and ")" mark the beginning and end of the argument list.

The **file_type_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file_type_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **object_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object_type** equals DISPLAYBUS, then the data describes DISPLAY BUS objects. Below is a list of many of the object types recognized by Simulator:

CaseInformationMemo	DisplayLineFlowArrow	DisplayTransmissionLineField
ColorMap	DisplayLoad	DisplayZone
Contour	DisplayLoadField	DisplayZoneField
CustomColors	DisplayMultiSectionLine	DisplayZonePie
DisplayArea	DisplayMultiSectionLineField	DocumentLink
DisplayAreaField	DisplayMultiSectionLinePie	DynamicFormatting
DisplayAreaPie	DisplayOwner	Ellipse
DisplayBranchGauge	DisplayOwnerField	Filter
DisplayBranchPie	DisplayOwnerPie	GeoDataViewStyle
DisplayBus	DisplaySeriesCapacitor	GeographyDisplayOptions
DisplayBusField	DisplaySeriesCapacitorField	Group
DisplayBusPie	DisplayShunt	Line
DisplayCircuitBreaker	DisplayShuntField	OnlineField
DisplayDCTransmissionLine	DisplayShuntPie	OnlineLink
DisplayDCTransmissionLineField	DisplaySubstation	Picture
DisplayGen	DisplaySubstationField	PieChartGaugeStyle
DisplayGenericModelField	DisplaySubstationPie	PWFormOptions
DisplayGenField	DisplaySuperArea	Rectangle
DisplayGenPie	DisplaySuperAreaField	Screenlayer
DisplayInjectionGroup	DisplaySuperAreaPie	SelectByCriteriaSet
DisplayInjectionGroupPie	DisplayTransformer	Text
DisplayInterface	DisplayTransformerField	TextBox
DisplayInterfaceField	DisplayTransformerPie	View
DisplayInterfacePie	DisplayTransmissionLine	

The list of object types Simulator's auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available

object_types by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**.

The **list_of_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list_of_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 3,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list_of_fields** are:

- The **list_of_fields** may take up several lines of the text file.
- The **list_of_fields** should be encompassed by braces [].
- When encountering the PowerWorld comment string `'''` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'''` will be ignored as comments.
- Field names must be separated by commas.

Example DATA Argument List

```
DATA (DISPLAYBUS, [BusNom, SOAuxiliaryID, // comment here
SOX, SOY, SOTickness, SOColor, SOUseFillColor, SOFillColor,
// comments allowed here too

// note that blank rows are ignored
SOSize, SOWidth, SOOrientation, SOLevel, SOImmobile,
SLName, SOSStyle, SODashed, // more comments
SOBelongsToGroup])
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field `BusNum` when identifying branches. Bus numbers must be used to identify the from and the to end of branches. To keep the number of fields from becoming too large, the same field name is used for both of these values. The from bus number is written as `BusNum:0` and the to bus number is written as `BusNum:1`. Note that the `:0` may be left off of field names.

The **create_if_not_found** field is optional. This specifies whether or not to create a new object if an existing one is not found. If the value is YES, objects will be created. If the value is NO, objects will not be created. If omitted, the default behavior of prompting the user about whether or not to create a non-existing object will continue. If loading and auxiliary file using the **LoadAxd** script command, the **create_if_not_found** field for the data section will override the **CreateIfNotFound** field with the script.

Data Key Fields for Display Auxiliary File

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for DISPLAYBUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for DISPLAYGEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list_of_fields** (i.e. they need not be the first fields listed in **list_of_fields**). As long as the key fields are present, Simulator can identify the specific object.

Display objects have an additional key field used for identification because multiple objects can be present on the same one-line diagram that represent the same power system element. This extra key field is SOAuxiliaryID. This is a field that is unique for each type of display object and other key field combination. If there are two display buses that represent bus one in the power system, the SOAuxiliaryID field will be different for both. Simulator will automatically create unique identifiers when these objects are created graphically. They can also be user specified but are forced to be unique. This field does not need to be present when reading in a display auxiliary file, but if it is missing, Simulator assumes that the ID is "1". This field is the only key field identifier for objects that do not link to power system elements such as background lines and pictures, and therefore, should always be included when reading in these objects or the expected results may not be achieved.

By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks *.

Data List for Display Auxiliary File

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list_of_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value_list**:

- The **value_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data_list**.
- If the **file_type_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.
- A string in the format `&[ModelExpressionName]:[#digits]:[#decimals]` will be treated as though the value of the named model expression is entered in the field, with `[#digits]` total and `[#decimals]` digits to the right of the decimal point. This format may be used in case information displays as well as in AUX files.

Special Data Sections

There are several object types that should be noted here because they can impact the reading of an entire display auxiliary file, overall look of the resulting one-line diagram, or require special input to properly import/export the object.

GeographyDisplayOptions

Most objects supported in the display auxiliary file have coordinates that can be specified in the appropriate data sections. What these coordinates specify can be controlled by the GEOGRAPHYDISPLAYOPTIONS object. This object has only two fields available: MapProjection and ShowLonLat. There are three possible settings for MapProjection: "x,y", "Simple Conic", and "Mercator". The choice of projection will determine how the x,y values for display objects are interpreted. ShowLonLat can be either "YES" or "NO". If ShowLonLat is "YES", the setting specified for the MapProjection will be the longitude,latitude projection used when reading/writing the object x,y values. If ShowLonLat is "NO", the x,y values will always be interpreted as x,y regardless of the MapProjection setting. This object should be placed in the display auxiliary file before any other objects containing coordinates are read. If this object is not included in the auxiliary file, the coordinates will be interpreted based on the current settings of map projection and whether or not coordinates are showing longitude,latitude.

Picture

PICTURE objects represent background images that cannot be stored in a text file format. To properly include a PICTURE object in a display auxiliary file, the file containing the image must be saved and read along with the auxiliary file. The FileName field indicates the name and location of the image file. If the image file cannot be found when reading in a display auxiliary file and attempting to create a new object, no PICTURE object will be created. If attempting to update an existing object and the image file cannot be found, the object will not be updated with a new image, but the FileName field will be updated with the specified file name.

PWFormOptions

One-line display options that affect the current display settings can be changed by using the PWFORMOPTIONS object. Usually, this object specifies named sets of options that can be selected and used to change the various one-line display options through the GUI. By including a specially named object, the current options can be changed through a display auxiliary file.

PWFORMOPTIONS are named using the OOName field. Setting this field to "THESE_OPTIONS_ARE_APPLIED_TO_THE_CURRENT_DISPLAY" will apply the specified set of options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a PWFORMOPTIONS object with this name is added to the file by default.

View

Different views can be specified in the display auxiliary file using the VIEW object. Usually, this object is used to specify named sets of options used to select and change the view through the GUI. By including a specially named object, the current view can be changed through a display auxiliary file. VIEW objects are named using the ViewName field. Setting this field to

"THIS_VIEW_IS_APPLIED_TO_THE_CURRENT_DISPLAY" will apply the specified set of view options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a VIEW object with this name is added to the file by default. If no view is specified as such during loading, the display defaults to a Show Full view.

Data Subdata Sections for Display Auxiliary File

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<SubData>`, `</SubData>` tags that store lists of information about a particular object. This formatting looks like the following:

DATA (object_type, [list_of_fields], file_type_specifier)

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
.
.
.
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.

Data Common Sections

ColorMap

Same format as in *data auxiliary* files.

CustomColors

Same format as in *data auxiliary* files.

DynamicFormatting

Same format as in *data auxiliary* files.

Filter

Same format as in *data auxiliary* files.

GeoDataViewStyle

Same format as in *data auxiliary* files.

PWFormOptions

Same format as in *data auxiliary* files.

SelectByCriteriaSet

Same format as in *data auxiliary* files.

UserDefinedDataGrid

Same format as in *data auxiliary* files.

Data DisplayLine

DisplayDCTramisssionLine

DisplayInterface

DisplayMultiSectionLine

DisplaySeriesCapacitor

DisplayTransformer

DisplayTransmissionLine

Line

This is a list of points defining the graphical line used to represent the object. Each set of coordinates can be enclosed in square brackets, [], or the brackets can be eliminated. The brackets will be included when Simulator generates an auxiliary file. The individual coordinates are separated by the specified delimiter, either a space or a comma, and if the brackets are included, the same delimiter should be used to separate sets of coordinates. The list of points is in a somewhat free form and sets of coordinates can span multiple lines. Each point should either be in x,y coordinates or longitude,latitude coordinates. Which coordinates should be used depends on the current option settings for map projection and whether or not coordinates should be shown in longitude,latitude. If the display auxiliary file is automatically generated by Simulator, a comment will be included in the subdata section indicating the coordinate system in use during file creation.

Example using brackets and a comma delimiter:

```
<SUBDATA Line>
//Coordinates are x,y
[14.00000000, 63.00000000], [14.00000000, 60.00000000],
[20.00000000, 45.00000000], [20.00000000, 42.00000000]
</SUBDATA>
```

Example with no brackets and a space delimiter:

```
<SUBDATA Line>
//Coordinates are x,y
14.00000000 63.00000000 14.00000000 60.00000000
20.00000000 45.00000000 20.00000000 42.00000000
</SUBDATA>
```

Data PieChartGaugeStyle

ColorMap

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

Percentage: This is the percentage at which the following scalar and color should be applied.

Scalar: A scalar (multiplier) on the size of the pie chart/gauge.

Color: A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>
//Percentage Scalar Color
85.0000 1.5000 33023
100.0000 2.0000 255
</SUBDATA>
```

Data View

ScreenLayer

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

Example:

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

Simulator Options

Simulator provides a flexible environment for simulating power system operation by offering you access to a number of customizable options. The **Simulator Options Dialog** houses pages of options that you can customize to tailor the program to your needs.

To display the Simulator Options Dialog go to the [Options](#) ribbon tab, and choose **Simulator Options** from the Case Options Ribbon Group. The Simulator Options button is also available on the [Tools](#) ribbon tab on the [Power Flow Tools](#) ribbon group.

If a timed simulation is active when you open the dialog, it is automatically paused and will not resume until you close the dialog.

There are seven categories of options for the Simulator Options dialog:

[Case Information Display Options](#)

[Environment Options](#)

[File Management Options](#)

[Message Log Options](#)

[Online Options](#)

[Distributed Computing Options](#)

[Power Flow Solution Options](#)

Power Flow Solution Options

The Power Flow Solution Tab offers various options regarding how Simulator solves the power flow problem. There are six sub-categories on the Power Flow Solutions tab: [Common Options](#), [Advanced Options](#), [Island-Based AGC](#), [DC Options](#), [General](#), and [Storage](#). Each of these categories is shown on tabs. Many of the options in the Power Flow Solution options will be of interest only to advanced users of the package.

Power Flow Solution: Common Options

The following options are all contained on the Common Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Power Flow (Inner) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

MVA Convergence Tolerance

The MVA convergence tolerance serves as a measure for determining when the inner power flow loop of the Power Flow Solution process has reached an acceptable solution. The MVA mismatch is computed as the maximum real or reactive mismatch at any bus in the system. Usually, this value should be around 0.1 MVA. If you are having difficulty solving a particular case, it may be helpful to temporarily increase the MVA Convergence Tolerance to drive the solution closer to the actual solution, and then re-solve from this solution using the smaller MVA tolerance.

Maximum Number of Iterations

This option defines the maximum number of iterations Simulator will perform during the Power Flow Solution process in an effort to converge to a solution. If Simulator must exceed this number of iterations, it assumes that the power flow case is not converging and will terminate the solution process. If Simulator is configured to represent non-converging power flow cases as blackouts, the screen will turn gray and the blackout warning message will appear.

Do Only One Iteration

If checked, then Simulator will only perform one iteration of the load flow solution process when **Single Solution** is clicked, regardless of the Maximum Number of Iterations setting. This is useful when playing with a case with which you are experiencing difficulty.

MW Control (Outer) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Automatic Generation Control (AGC)

If checked, the enforcement of the Generation Re-dispatch to account for MW interchange constraints for all areas is disabled. By default, this option is not checked.

Enforce Generator MW Limits

If checked, then generator minimum and maximum MW limits are enforced for all generators whose *Enforce MW Limits* field is set to true. See [Generator Information Dialog](#) for more information. Otherwise, generator MW limits are not enforced. Note that when using the economic dispatch or the Optimal Power Flow, the generator MW limits are always enforced regardless of user-settings.

Topology Processing Options (These options are only available with the [Topology Processing](#) add-on)

Use Topology Processing

This option is used for all Simulator applications: power flow, contingency analysis, sensitivity calculations, etc. This option should ALWAYS be checked for real-time cases, while it should be unchecked for planning cases. If a full-topology model was read and this option is not checked, the power flow will try to solve the model without removing the switching devices resulting in a very large [ill-conditioned Jacobian](#). This option is also found on the [Topology Processing Dialog](#).

Close Breakers to Energize Switched Shunts

If checked, an attempt will be made to close breakers in order to energize a [Switched Shunt](#) that is on either Discrete or Continuous control and is needed to meet its regulated value. This option is applied as part of the [switched shunt control](#) any time that the power flow is solved.

Controller (Middle) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Checking Gen VAR Limits

If checked, the Mvar limits are ignored for all the generators in the case during a Power Flow Solution. By default, this option is not selected.

Check Immediately

Before entering the inner power flow loop to solve the power flow equations, a decision is made about each generator in the case as to whether it will be treated as a PV or a PQ bus. It is assumed that a PV bus will maintain its setpoint voltage, while a PQ bus will maintain a constant reactive power output (for instance when the generator is at a Mvar min or max limit). Normally, the power flow equations are completely solved in an inner power flow loop using this assumption. After completing the solution, generator Mvar limits are checked to see if they have been reached (must change from PV to PQ), or if at a limit if the limit may be backed off of (must change from PQ to PV). If any generators change PQ/PV status, then the inner power flow loop is resolved.

When the **Check Immediately** option is checked, the generator Mvar limits are checked after *every* inner power flow loop iteration instead to see if generator limits have been reached, or if a generator limit may be backed off. When this option is not checked then it will not check the generator limits inside the inner power flow loop. In some situations performing this check inside the inner power flow loop can help with convergence, however performing this check does slow down the solution process. By default, this option is not selected.

Disable Switched Shunt (SS) Control

If checked, automatic control of switched shunts is disabled in all areas. By default, this option is not selected.

Disable LTC Transformer Control

If checked, automatic control of LTC transformers is disabled in all areas. By default, this option is not checked.

Disable Phase Shifter Transformer Control

If checked, then automatic control of phase shifting transformers is disabled in all areas. By default, this option is not checked.

Transformer Stepping Methodology

Choose either *Coordinated Sensitivities* or *Self-Sensitivity Only*. The default value is *Coordinated Sensitivities*.

When the regulated value of tap-changing or phase-shifting transformers move outside of their regulation range, then Simulator attempts to bring those transformers back inside their range. By default Simulator determines all the tap-changing and phase-shifter transformers which are out-of-range and coordinates the movement of these transformers in an attempt to bring all the regulated values back into range. This is what is called *Coordinated Sensitivities*. Generally this results in a better convergence, however it also can be slower when a large number of transformers are out-of-range together. Because it gets slower with a large number of transforms, if more than 50 transformers are involved in the switching, Simulator will always use the *Self-*

Sensitivity Only option. As the solution process continues however, the number of transformers moving will reduce and eventually the *Coordinated Sensitivities* will be used again.

Choosing *Self-Sensitivity Only* will modify this methodology so that each transformer only looks at the sensitivity of its regulated value with respect to changing its own tap or phase. This calculation is faster, but may result in convergence problems due to transformers that interact with one another.

Prevent Controller Oscillations

Sometimes, a power flow will fail to converge because certain automatic controls such as Mvar limit enforcement at generators, transformer tap switching, and shunt switching oscillate between their control bounds. These oscillations very often are due to modeling inaccuracies. If this option is checked, Simulator will automatically detect such oscillating controls and fix them at their current value so that they no longer oscillate. You may find this option helpful if you feel that the modeling of automatic controls in your system is inaccurate.

Disable Transformer Tap Control of a Terminal Bus if Tap Sensitivity is the Wrong Sign (Normally Check This)

As part of the solution process, if the transformer is regulating one of their own terminal buses then simulator usually disable the transformer tap control of a terminal bus if the tap sensitivity detect that the regulated bus will move the voltage to the opposite direction. If this option is unchecked then simulator still will allow the transformer tap control even is the tap sensitivity detect that the regulated bus will move the voltage to the opposite direction. It is recommended to always lives this option checked.

Maximum Number Controller Loop Iterations

As part of the solution process, the outer loop of the solution algorithm is a check of any necessary controller changes due to changes in controlled values from the last iteration of the Newton-Raphson load flow solution. The maximum number of loops through the control change algorithm can be set here. This is not the same as the Maximum Number of Iterations, which applies to the actual Newton-Raphson inner loop algorithm, which solves that actual power flow.

Power Flow Solution: Advanced Options

The following options are all contained on the Advanced Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Dynamically add/remove slack buses as topology is changed

If checked, Simulator dynamically chooses a new slack bus for the islands if the user-input does not adequately define slack buses. If a slack bus can not be determined for an island, the island cannot be solved and will be isolated and ignored during the load flow solution. Also if there are multiple slack buses specified in the same island, Simulator will choose only one of them as the slack bus.

When allowing dynamic slack bus assignment, the choice of the island slack bus is managed in Simulator by internally maintaining a list of buses the user has identified as slack buses while in Edit Mode. In Run Mode, electrical islands are then dynamically determined from the system topology. As branch statuses are changed, this list of islands (and slack buses) will automatically change. An island will be considered viable only if it meets the following criteria.

1. At least one island bus has some on-line generation with a positive Maximum MW Limit and is NOT set to regulate the voltage at a bus in another island.
2. At least one island bus has a closed load (Note: It could still be a 0.0 MW load) OR the island is connected to another island by a DC transmission line.

After the viability of each island is determined, a slack bus will be chosen for each viable island. (Note: appropriate messages are written to the log when buses are turned into slack buses or removed as slack buses). If an island has exactly one user-identified slack bus, then that is used as the slack. If an island has greater than one user-identified slack bus, then the user-identified slack bus which has the largest maximum generator MW output is used as the slack bus. If an island has no user-identified slack buses, then one of the buses in the island will be automatically chosen as the slack bus. When choosing this bus, the four criteria listed below are used. If buses are found that meets all four criteria, then of those buses the one with the largest maximum generator MW limit will be used. If no bus is found which meets these criteria, then of the buses that meet the first three criteria, the one with the largest maximum generator MW limit will be used. Again, if no bus is found, the first two criteria are used and so on. Under no circumstance will a generator set to any kind of wind power control mode be selected as an island slack bus.

1. At least one generator at the bus is set to AVR = YES
2. The sum of the Maximum MW limits for generators at the bus is less than 5,000 MW (if there is more, these are probably a fictitious MW limits)
3. Generators at the bus regulate their terminal bus (this will be required by the slack bus anyway). However, if another bus with generation that does not regulate its terminal bus is available and this other bus has a maximum MW output that is 5 times higher, it will be chosen.
4. There is only one generator at the bus (it is just easier for the user to keep track of then)

Post Power Flow Solution Actions

Clicking this option will open the [Post Power Flow Solution Actions dialog](#), where the user can specify a list of actions to be executed at the end of every Full AC power flow solution.

Power Flow (Inner) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Power Flow Optimal Multiplier

If checked, the Newton solution process will not use the optimal multiplier. The optimal multiplier is a mathematically calculated step size for the Newton's Method iteration that prevents the mismatch equations from increasing between iterations. A small value indicates that moving in the direction of the Newton step will increase the mismatch equations. When the optimal multiplier becomes too small, this is an indication that the power flow is not going to solve successfully and Simulator will stop instead of creating extremely large mismatches. If this happens at a point where the solution is not within the allowed tolerance for the Newton process, the Newton process will result in a failed convergence to a valid solution.

Initialize From Flat Start Values

This option only applies to stand alone power flow solutions. Stand alone means a power flow solution initiated by clicking on the Single Solution button on the [Power Flow Tools group](#) in the [Tools Ribbon Tab](#). This option does not apply to power flow solution initiated by the contingency analysis, PV Curve, QV Curve, or ATC tools.

When checked, each Power Flow Solution is started assuming that all voltage magnitudes at buses that are not being regulated by a generator or have attached generators are set to unity. If a bus is regulated by a generator or is a generator bus, the voltage magnitude is set to the voltage setpoint of the generator. Voltage angles are set to the voltage angle of the system slack bus. By default, this option is not selected. Some power flow problems can be very difficult to solve from flat start assumptions. Therefore, use this option sparingly.

Minimum Per Unit Voltage for Constant Current Loads

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.5.

For constant current loads with a terminal bus below this per unit voltage, the value of the load will fall off using a sine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

Minimum Per Unit Voltage for Constant Power Loads

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.7.

For constant power loads with a terminal bus below this per unit voltage, the value of the load will fall off using a cosine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

Control (Middle) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

Disable Treating Continuous Switched Shunts (SSs) as PV Buses

Continuous switched shunts are normally treated the same as a generator bus inside the inner power flow loop (they are treated as buses with fixed power and voltage). Checking this option will cause the continuous switched shunt to be treated as a constant impedance in the inner power flow loop with all switch occurring in conjunction with other switched shunts in the voltage control loop instead.

Disable Balancing of Parallel LTC Taps

Simulator has the capability to attempt to balance tap positions of parallel transformers, in an attempt to avoid parallel transformers from going to opposite tap settings, inducing loop flow through the parallel transformers. Checking this option disables the automatic balancing of parallel transformers. The only transformers that will be balanced are those in parallel between the same

terminal buses, or those in parallel between terminal buses that are connected with very-low impedance branches.

Model Phase Shifters as Discrete Controls

If checked, then phase shifters will switch tap positions discretely based on the tap step size of the phase shifting transformer. By default, this option is not checked, which means the phase shifters will switch continuously, independent of the tap step size.

Minimum Sensitivity for LTC Control

This option specifies the minimum-voltage-to-tap sensitivity for LTC transformers. All transformers having an absolute value of voltage-to-tap sensitivity below this value are automatically disabled from automatic control. This prevents Simulator from changing transformer taps that have little effect on their controlling voltage. The [Transformer AVR Dialog](#) shows the voltage-to-tap sensitivity for each voltage-controlling transformer.

Disable Angle Smoothing

When a transmission branch status is changed from OPEN to CLOSED across a branch that has a large voltage angle difference, this can introduce a very large initial power flow mismatch and cause the inner power flow loop to diverge. Angle Smoothing will alleviate this large angle difference by smoothing the angles in the system around the newly closed in branch resulting in much better power flow convergence. Angle smoothing will also work if a series of branches are all closed in together. Angle smoothing should be enabled by default, but checking this option will disable this. Angle smoothing works best for individual branches or a series of branches that have been closed that are not electrically near other branches that have been closed. If modifying a case and adding in new transmission lines that are electrically near each other, it might be better for solution convergence to disable angle smoothing.

Disable Angle Rotation Processing

At the end of a power flow solution, by Simulator looks at the bus voltage angles in the case to see if any are near +/- 180 degrees. If the angles fall outside of +/- 160 degrees, then all angles in the island will be rotated by the same amount so that the angle range in the islands is equally spaced around zero degrees. Check this option to disable this feature.

Sharing of generator Vars across groups of buses during remote regulation

When several buses have generators that control the voltage at single bus, this option determines the method used to determine each buses "share" of the MVar support. There are three options

1. Allocate across buses using the user-specified remote regulation percentages. This option is what is used by default and most closely matches the sharing seen in RAW files.
2. Allocate so all generators are at same relative point in their [min .. max] var range. This option most closely matches the sharing seen in a few EMS solutions PowerWorld has seen.
3. Allocate across buses using the SUM OF user-specified remote regulation percentages. This option most closely matches the sharing seen in EPC files.

It should be noted, generators at the same bus always allocate vars so they are at the same relative point in their [min..max] MVar range. How vars are allocated to generators at the same bus has no effect outside of that bus so options are not given for this sharing.

Include Loss Penalty Factors in ED

If checked, the economic dispatch calculation will consider losses in determining the most economic generation dispatch. Otherwise, the generation dispatch calculation will disregard system losses.

Enforce Convex Cost Curves in ED

The economic dispatch algorithm attempts to set the output of all generators that are set to be automatically controlled so that the system's load, losses, and interchange are met as economically as possible. The algorithm is guaranteed to reach a unique solution only when all generator cost curves, which model the variation of the cost of operating a unit with its output, are convex. If this option is checked, Simulator will identify units whose operating point is outside the convex portion of the cost curve and set them off automatic control.

Post Power Flow Solution Actions Dialog

The Post Power Flow Solution Actions dialog describes a list of actions that are executed at the end of every Full AC power flow solution, which means they are not performed for DC solutions. Normally, these actions would all have Model criteria specified.

To open this dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. Next, go to the **Power Flow Solution** category. Then click on the **Advanced Options**. Finally click the button labeled **Define Post Power Flow Solution Actions**.

Check the **Do Not Used Post Power Flow Solution Action List** check-box to disable using any actions defined in the list.

The action list display identifies the actions that comprise the post power flow solution action list. Actions can be inserted or deleted by using the local menu on the dialog. Right-click on the display and select **Insert** or **Delete**. Actions are inserted via the [Contingency Element Dialog](#).

The action display is a type of [Case Information Display](#) and thus shares many common to all other case information displays.

The Action List Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the actions by right-clicking on the Action List Display and choosing **Display Descriptions By**, and then choosing either Name, Num, Name/Num, PW File Format by Numbers, PW File Format by Name/kV or PTI File Format.

Model Criteria

Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). These specify a criteria under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the Model Condition and Filter option on the [Contingency Options Tab](#). You can open a dialog to define Model Filters or Conditions by right-clicking on the Action List Display and choosing **Define Model Criteria**.

Status

Three options are available for this field

- **CHECK** : The action will be executed only if the Model Criteria is true. It will also be executed if no model criteria is specified. This is the default setting
- **ALWAYS**: The action will always be executed, regardless of the Model Criteria.
- **NEVER**: The action will never be executed, regardless of the Model Criteria.
- **POSTCHECK**: This action is checked AFTER the other Check and Always actions have been performed and the load flow solution solved. If the criteria specified for the Postcheck action are met in the resulting load flow solution, then this action is taken and the load flow is again resolved. This will recursively occur for all Postcheck actions until either all postcheck actions have been taken, or the criteria for all remaining postcheck actions have not been met.

Note that the Never action allows you to disable a particular action without deleting it.

Comment

A user-specified comment string that can be associated with this action. While this comment is not used by Simulator in any way, these comments can be saved to be loaded at a later time. This is provided for the user to add comments regarding the action. For example, for an action with a Model Criteria you may could add a sentence explaining why the action is only performed under the specified criteria.

Power Flow Solution: Island-Based AGC

The following options are all contained on the Island-Based AGC tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

The Island Based AGC options allow the user to choose to dispatch generation by island instead of by area or super area. The options available for dispatch here are:

Disable (Use the Area and Super Area Dispatch Settings)

When this option is selected, Island Based AGC is being used, and generators are dispatched according to the usual area or super area generation dispatch.

Use Participation Factors of individual generators

When selected, the island-based AGC is used, based on the individual participation factors of each generator within the island. Area and super area ACE requirements will be ignored, and all generators in the island will be dispatched to serve load, losses, and any DC transfers to other islands.

Note that if all generators with AGC = YES have participation factors also equal to 0.0, then all make-up power will come from the island slack bus.

Calculate Participation Factors from Area Make Up Power Values

The AGC will dispatch generation by use area make up power values, instead of by individual participation factor. What this means is that each area will be assigned a make up power value, similar to assigning a participation factor to a generator. Based on the make up power value of each area in the island, the amount of the generation dispatch needed will be divided amongst each area based on its value. Higher value will account for more of the generation dispatch than areas with smaller values. The within each area, the generation dispatch is handled on an individual generator participation factor basis, where each generator will account for a portion of the dispatch that was assigned to its area.

To specify the make up power values for each area, click on the **Specify Area Make Up Power Values** button. Note that these settings are the same as those used by [Contingency Analysis Options Make Up Power](#), therefore changing these values will affect the contingency analysis.

For example, three areas have participation factors of 2, 1 and 1, respectively. If the total generation redispatch in the island is 100 MW, then area 1 will account for $2 / (2+1+1)$, or 50%, of the total. Therefore area 1 is expected to redispatch by 50 MW. If area 1 then has two generators with participation factors of 4 and 1, they will account for the 50 MW by picking up $4 / (4+1)$ and $1 / (4+1)$, or 80% and 20%, respectively, of the 50 MW needed from the area.

Note that if all areas have **CTG Make Up Gen** equal to 0.0, then all make-up power will come from the island slack bus.

Dispatch using an Injection Group (Loads and Generators will respond)

Checking this option will allow for the island dispatch to be covered by change in generation and/or load defined in an [injection group](#).

Additional options are available when dispatching based on an injection group:

Allow only AGC Units to Vary

If this option is checked, only units whose AGC status is turned on will be allowed to participate in the injection group dispatch.

Enforce unit MW limits

If checked, then each generator's defined MW limits will be strictly adhered to during the redispatch.

Do not allow negative loads

When checked, loads included in the injection group are not allowed to drop below zero MW or MVAR load demand.

How should reactive power load change as real power load is ramped?

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

Island AGC Tolerance

This sets the AGC tolerance in MW for the real power balance of each island for all options except **Disable (Use the Area and Super Area Dispatch settings)**. Generation must equal load plus losses plus DC line exports in each island. If necessary, each island slack bus output will vary during the [Power Flow \(Inner\) Loop](#) solution to maintain the real power balance for the island. If it varies more than the specified tolerance and if a form of Island-based AGC is specified, then adjustments to AGCable units will be made in the [MW Control \(Outer\) Loop](#) according to the selected Island-based AGC option.

Power Flow Solution: DC Options

The following options are all contained on the DC Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Use DC Approximation in Power Flow

When this box is checked, Simulator will solve the load flow using the DC Approximation method. When not checked, Simulator performs the modified Full Newton AC load flow algorithm. Note that once a case, especially a large one, has been solved using a DC approximation, it tends to be quite difficult to revert back to the Full Newton AC load flow from a solved DC approximation.

DC Power Flow Model

Note: These choices also affect the calculation of sensitivities on the [PTDF](#), [LODF](#), and [TLR](#) dialogs and [ATC Tool](#) calculations.

Generally, the *only* inputs that affects the DC Power Flow equations are line impedances and line status (open/closed). However there are three possible modeling choices (for which there is not general agreement in the industry) that will affect the "DC equations" in small ways. These choices are made here on the DC Power Flow Model options

Ignore Series Resistance (r) or Ignore Series Conductance (g)

The series term of a transmission line consists of an r and x value which represent a complex number impedance. In the DC power flow approximation, the value that used is the imaginary part of the inverse of impedance (called admittance)

$$g + jb = 1 / (r + jx) = r / (r^2 + x^2) - j x / (r^2 + x^2)$$

Thus

$$g = r / (r^2 + x^2)$$

$$b = - x / (r^2 + x^2) \quad *** \text{ This term is used in the DC power flow equations}$$

The only term that is used in the DC power flow equations is the b term. Some say that a DC power flow means that r = 0 which means that b = -1/x and g = 0. Others say that a DC power flow only means assuming that g = 0, which means that b = - x / (r^2 + x^2). There is not a good consensus in the industry as to what is correct regarding this, so Simulator offers an option as to whether you ignore r or ignore g. The default option in PowerWorld is to ignore the resistance.

Ignore Transformer Impedance Correction Tables and Ignore Phase Shift Angle Effects

These are both ignored by default.

When a transformer impedance correction table is specified for a transformer, then the series impedance of the transformer will vary by a multiplier with the tap or phase of the transformer. The tap ratio is not generally relevant to a DC power flow, however the phase-shifter angle could vary when using the DC approximation. This option specifies whether to ignore the impact of this impedance correction. For phase shifters the multiplier is normally between 1.00 and 1.50 and thus tends to *increase* the impedances as you move away from zero degrees.

As a branches phase shift varies, the DC power flow equations could be modified to take this into account. This effect results in the DC power flow equations for which the "b" term above is multiplied by the cosine of the phase angle across the line. This effect tends to *decrease* the impedance as you move away from zero degrees.

Because the impedance correction tends to increase impedance and the phase effect tends to decrease impedance it is reasonable to ignore both of these, and this is done by default. The other issue with not ignoring these is that both create a situation where the equations for the DC

approximation are dependent on the system state. In other words, the matrix equations become a function of the phase-shift angle. This results in a situation in which the matrices must be recalculated and re-inverted each time a phase angle is moved. This removes some of the advantages of the DC approximation. This can be especially problematic when using the OPF or SCOPF.

Compensate for Losses by Adjusting the Load

Traditionally a DC load flow is treated as lossless. However, you can approximate the loss in the load flow by artificially adjusting the load in the case to include estimated losses. To do so, click on the **DC Loss Setup** button to open the [DC Power Flow Loss Setup](#) dialog for setting the DC Loss Multipliers.

Compensate for Reactive Power Flows by Adjusting the Branch Limits ...

One issue with DC load flow is how to treat the inclusion of reactive power flows. In a standard DC load flow, the reactive flows are typically ignored, and the MW flow of a branch is compared to its original MVA branch limit. However, Simulator also gives you the option to instead solve the DC load flow and compute approximate MVAR flows by assuming the voltages in the system remain constant when the DC load flow is solved.

Compensate for Dispatch Sensitivities with User-Specified Values

This option allows for the bus MW loss sensitivities to be used in the OPF and ED dispatch algorithms, if the type of loss sensitivity on the [General](#) tab is set to User-Specified.

DC Power Flow Loss Setup

To open the DC Power Flow Loss Setup dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. On the Power Flow Solution page, click on the tab labeled **DC Options**. Next, click the button labeled **DC Loss Setup**. This button will only be enabled when you check the box to **Use DC Approximation in Power Flow/OPF/SCOPF**.

The DC Power Flow Loss Setup dialog gives you a location to apply approximate losses during a DC load flow solution. The losses can be approximated by scaling the loads in the case to include an approximation of losses. Loss multiplication factors can be applied individually by bus, or as a group by area or by zone. Note that loss multipliers by area or by zone are just quick ways for setting the bus multiplication factors for all buses in the group. You will see the value reflected for all buses in the Buses page. If you wish to apply the same multiplication factor to the entire case, you can simply set the Case DC Loss Multiplier at the bottom of the dialog. This will automatically set all buses in the case to have the same DC Loss Multiplier specified.

Once the DC Loss Multipliers have been set, click OK to save the multipliers.

Power Flow Solution: General

The following options are all contained on the General tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Assumed MVA Per-Unit Base

This option allows the user to specify the MVA base to be used for the entire case. By default, this value is set to 100 MVA. Click on the **Change System Base** button to change this value and then update all the internal structures to store the proper values on the new base.

Bus Loss Sensitivity Function

Bus loss sensitivities indicate how island or area losses change with power injection at the bus. Here you may choose to forego the calculation of bus loss sensitivities or to base them on island losses or area losses. If the case consists of only one island, which, by definition, corresponds to the entire system, then the bus loss sensitivities are measured with respect to total system losses. If the bus sensitivities are set to User-Specified, the sensitivities will remain at their last calculated values, according to the loss function type previously specified when the [loss sensitivities](#) were calculated.

Monitor/Enforce Contingent Interface Elements

This global location allows you to determine how contingency elements in an interface should be treated in Simulator. You can choose to never include the impact of contingent elements on interface flow, to only include contingent element impacts in the standard power flow or optimal power flow routines, or in all solution routines including contingency analysis and security constrained OPF.

It is not uncommon to ignore the impact of contingent elements when using the contingency analysis or security constrained OPF tools, as they are already processing lists of contingencies and evaluating flows on interfaces. Ignoring contingent elements within interface definitions allows for a determination of the impact of other contingencies on the flows of the non-contingent elements forming the interface, without impact from additional contingent element considerations.

Power Units for Display

Allows switching between displaying power values in Mega- units or Kilo- units.

Power Flow Solution: Storage

The following options are all contained on the Storage tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Simulator offers the ability to restore either the last power flow solution state or the state of the system immediately before the last solution attempt. If your system has insufficient memory and you are working with large systems, you may wish to disable one or both of these options.

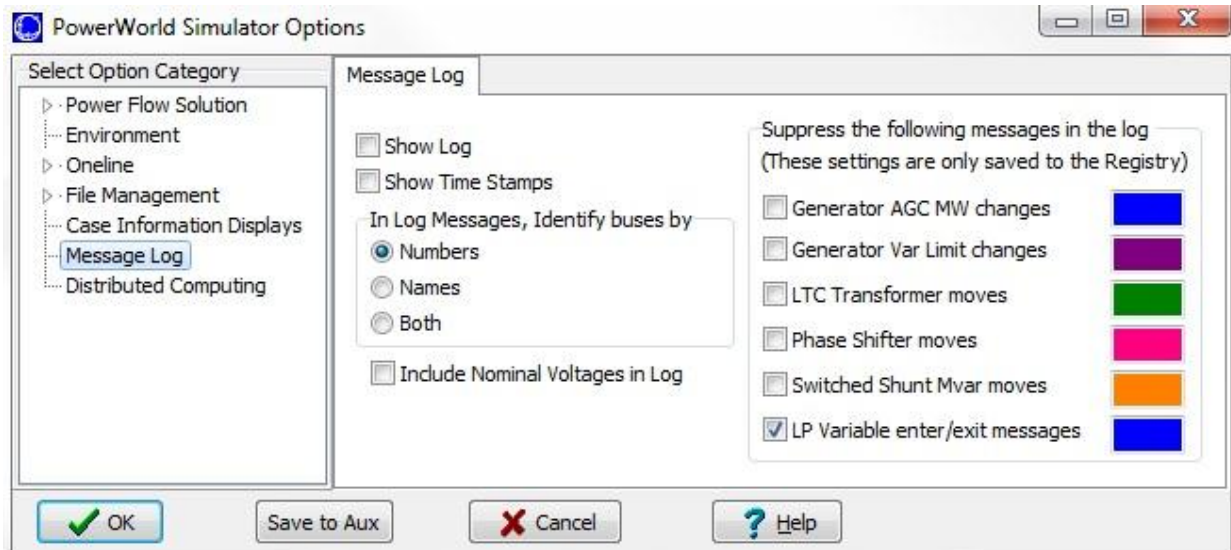
Disable "Restore last solution"

Restoring the last solution will undo any changes made to the data that were made after the last successful solution and return the case to the last valid solution. For large cases, the amount of memory required to store the last solution can be significant. If this option is checked, Simulator will not store this information in memory, and the last solution cannot be restored if a solution fails.

Disable "Restore state before failed solution attempt"

Restoring the state before failed solution attempt will undo only the attempted solution, but will retain any changes to data that were made before the solution process. This allows the user to return to the point just before the solution in order to add or remove changes in an effort to obtain a valid solution. For large cases, the amount of memory required to store the state before a solution attempt can be significant. If checked, Simulator will not store the state information in memory, and the state before the failed solution cannot be restored.

Message Log Options



Show Log

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

Show Time Stamps

Checking this option will display the date and time at which each message is posted in the message log.

In log Messages, Identify buses by

This option allows the user to specify how to identify the buses in log messages. The options are by numbers, by names, or by both numbers and names.

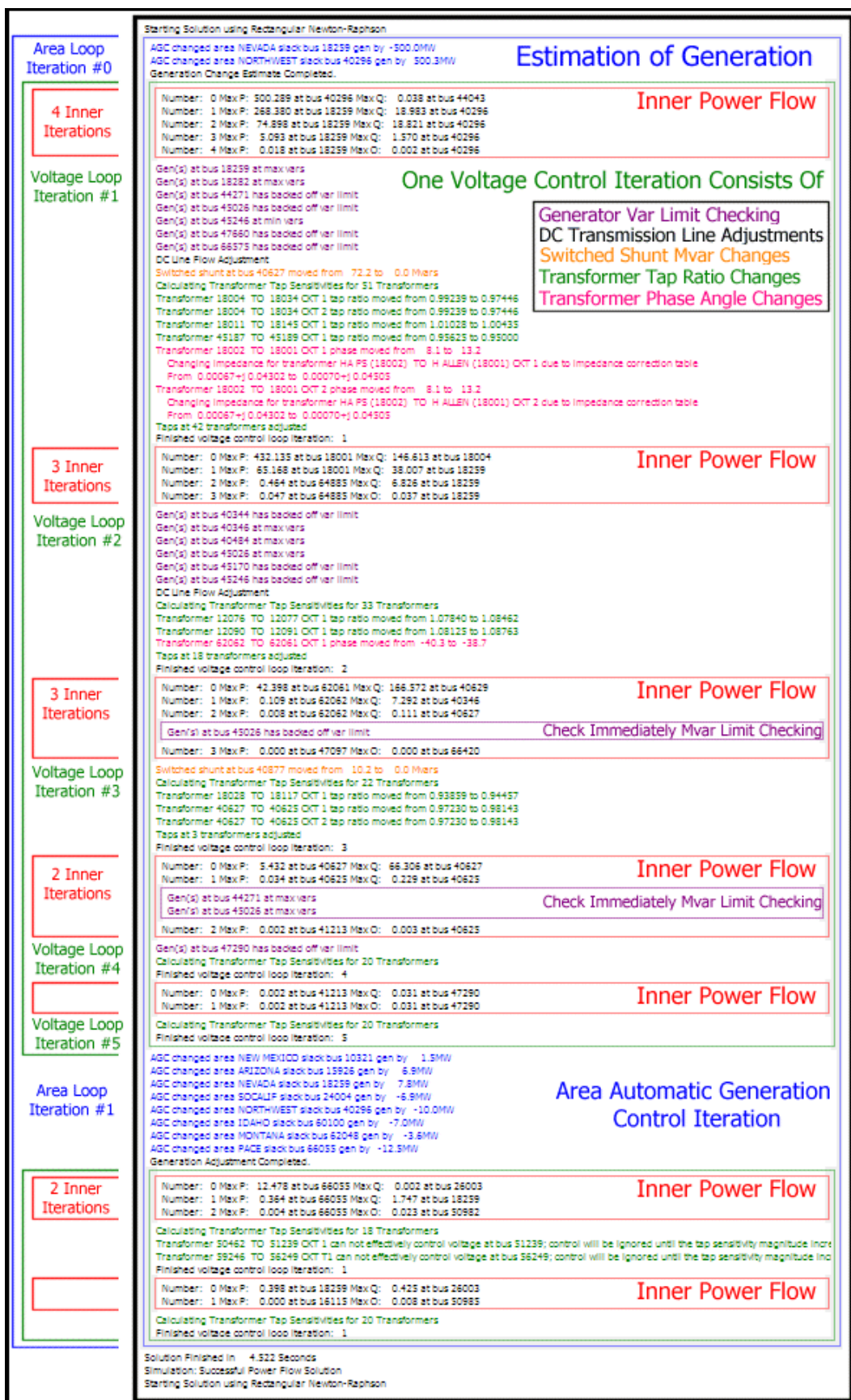
Include Nominal Voltages in Log

If checked, this option will make the buses to be displayed with their nominal voltages after their name.

Suppress the following messages in the log

Checking the check-boxes in this option will remove the corresponding message writing to the log, thus speeding up the computation process. The boxes to the right of the messages indicate the color with which the messages will be displayed in the log.

An example of how these colors are reflected in the message log is shown in the following image.



Environment Options

The Environment page provides you control over a number of display and simulation options. The first section of the page contains a list of check boxes that you can use to designate the content of the oneline displays.

These options include:

Do Not Solve While Animating (Display Only)

If checked, Simulator only displays the case; it does not solve the power flow equations. System flows are determined by the initial values in the case file. This option should be checked if you simply want to use Simulator to visualize a case that has already been solved. The advantage of the display-only mode is that animation is significantly faster, particularly for large cases. The drawback to the display-only mode is that the power flow equations are not automatically solved at each time step; you must explicitly call for a Power Flow Solution using either the **Single Solution** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab or choose one of the solution options under the **Solve** menu in the [Power Flow Tools](#) ribbon group.

Play/Animation Solution Method

This option is available only for OPF releases of Simulator. Use it to indicate whether Simulator's repetitive power flow should perform a normal power flow solution or an optimal power flow (OPF) solution.

Solution Animation

Check Auto Start Solution Animation to automatically start the animation after loading the case.

Check Auto Solve On Load to automatically solve the power flow solution when the case is loaded.

Show Log

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

Disable Showing Blackouts

You can dramatize a power flow case's failure to converge by representing it as a blackout. The background of the oneline diagram will become a dark shade of gray, and a message box will appear to announce that the system has experienced a blackout. To disable this behavior, select the Disable Showing Blackouts box. Representing the failure to converge as a blackout can be very helpful for presentation purposes. Very often, the power flow's failure to converge can be traced to the system's inability to serve the load demand, a situation that requires that load be "blackened out," or shed, to restore the system to a viable operating state. Thus, displaying the failure to converge as a blackout has physical significance.

Disable AGC When Manually Changing Generator MW

When this option is checked, changing the MW output of a generator manually will automatically remove a generator from [Automatic Generation Control](#). If you wish for generators to maintain their automatic generation control settings following a manual change of MW output, you must uncheck this option.

Open Associated Oneline Diagrams

By default, Simulator has always attempted to open any diagrams that were open with a case file when the case file was last closed. However, now you may uncheck this option and Simulator will not attempt to open any oneline diagrams when a case is loaded.

Auto Open Bus Records if No Oneline

If a case is opened in Simulator which does not have an associated oneline diagram, then Simulator will automatically open the [Bus Records](#) case information display if this option is checked.

Allow to Prompt for comment when saving case

If checked it will force saving the case with comments by popping a [dialog](#) every time a case is saved.

Auto Set Case as Base Case for Difference Flows when Loaded

This option allows the case to be set as a base case automatically after being loaded. This can be done always or never, or only if the base case has not been set before opening the case.

Reset Default Positions and Styles for all Dialogs

Click this button to reset the default position, size and Free-Floating/Contained style of the dialogs. See [Window Basics](#) for more information.

Custom Colors

This option allows the user to edit the custom colors used throughout all Simulator color dialogs.

Clock Style

The clock serves as a timer for timed simulations by showing the current time, the start time, and the end time of the simulation. You can choose to hide the simulation clock by specifying a clock style of *None*. Otherwise, to display the clock in its own window, choose *Dialog*, and to display the clock on the program's status bar, select *Status Bar*.

Measurement System

This option allows the user to choose English (Imperial) or Metric (SI) units for system measurements. By default, this option is set to English units.

Recently Used File List Entries

The maximum number of file names and locations stored in the History List of the File menu.

Undo Memory Limit Per Oneline

The approximate maximum number of Megabytes allowed to be used in storing actions carried out when editing a oneline diagram, which can be undone at any time.

Online Options

These options are available on the Online tab of the [PowerWorld Simulator Options](#) dialog. It is important to note that the options on this dialog apply to *all* online diagrams. There are also a large number of options which apply only to an individual online diagram. The diagram-specific options are specified on the [Online Display Options Dialog](#).

There are two sub-categories on the Online Options tab: Visualization and File.

Visualization

Visualizing out-of-service elements

These three options allow you to choose how objects on the diagram should appear when they are representing a power system device that is currently "open" or "out-of-service." The three options are Blink, Use dashed lines, and Draw and X through off-line generators. The first two options apply to any online object, while the third option is specific to generator objects only.

Set gen MW to min when closed from online

This option allows the generator MW value to be set to its minimum MW output after a circuit breaker is clicked in the online display. Normally the generator's status is set to closed and the MW value is reset to its previous value.

Show Online Hints

If checked, pop-up hints will appear when you drag the mouse over an object in the online. These give information about the object; for example, for a generator the pop-up hint displays the bus number, generator ID, the MW output and the MVAR output. You may also customize the pop-up hints for each kind of display object by choosing [Options > Custom Hint Values](#) from the menu.

Show X,Y Coordinates

If checked, the (x,y) location of the cursor is monitored in the status bar at the bottom of the screen. The (x,y) location of the cursor is only shown in Edit Mode. By default, this option is selected.

Save Contour Image with Online File

If checked, and if a contour is being displayed on the online diagram, Simulator will store the contour with the online diagram when you save the case or save the online. It will store the contour as a bitmap.

Display Unlinked Elements in Run Mode

Typically, unlinked graphical objects are not visible in Run Mode. Checking this option will allow the display of unlinked elements during Run Mode.

Enable Mouse Wheel Zooming

This option allow the use of the mouse wheel to zoom in and out in the currently selected online diagram.

Maximum number of open onelines

Option to specify the maximum number of onelines that can be open simultaneously (this option defaults to 10). If this maximum threshold is exceeded than PowerWorld will automatically close the online whose most recent access is the oldest. By default a online can be closed in this manner, but an additional online-specific option has been added that can be changed to prevent this closure. This feature is useful for users who make extensive use of online links or substation display object links which can make opening dozens of onelines very easy. Opening dozens of onelines can

slow down the program substantially. If you have a main oneline from which other onelines are automatically opened, then you may want to prevent the main oneline from closing.

Minimum Screen Font Size

The minimum font size at which text is visible on the screen. When text is rendered to the screen, any text which is smaller than this font size will not be rendered. This is useful when zooming out on a oneline diagram where a lot of text might become cluttered or hard to read on the screen.

Minimum Print/Copy Font Size

The minimum font size at which text can be printed or copied. This is useful if the application or printer you are sending to can or cannot display smaller fonts. When text is printed, any text which is smaller than this font size will not be printed. Normally the Minimum Print/Copy Font Size is smaller than the Minimum Screen Font Size because the resolution of the printed document is much higher than the screen resolution.

Transformer Symbol

Since transformer representation varies in different countries, this option allows the user to represent transformers as coils or circles. By default, transformers are represented as coils.

Line Navigation Arrows

These settings control whether the Pixel Size of the arrows, how long in seconds it should take to pan to the line end locations, and whether to show navigation hints as your mouse hovers over the arrow. See [Online Zoom and Panning](#) for more information.

File

Automatic Loading of Display Auxiliary File

This option is used options to automatically load a Display Auxiliary File (AXD) with one specific oneline. This setting is stored in the case (PWB) so that it only applies to onelines opened with the specific case. The file full path location has to be specified in the **Display Auxiliary File** box.

Automatic Loading of Display Auxiliary File with ANY oneline (Only saved to the Registry)

This option is used options to automatically load a Display Auxiliary File (AXD) with each oneline that is opened. This setting is stored in the windows registry so this applies to all onelines opened on the computer. The file full path location has to be specified in the **Display Auxiliary File** box.

Main Oneline File

This option is used to identify the primary oneline diagram to use with the case. The main oneline is the file that is displayed when you first open the case. The dropdown box lists all the oneline files that reside in the same directory as the case. Select one of these files, or enter the full path of the oneline you want to use if it does not appear in the dropdown box.

Use Default Oneline File

You can command Simulator to open a particular oneline diagram file if it cannot find a oneline diagram file for the case you are trying to open. For example, there is no oneline diagram associated with a PSS/E raw data file when you first read it into Simulator. However, if your application is such that you will always use the same oneline file whenever you open a PSS/E raw file, check the **Use Default Oneline File** to have Simulator open the oneline identified in the **Default Oneline File** box whenever it encounters a case that has no associated oneline. The default oneline must exist in the same directory as the case you are trying to open.

Automatically Show Full when opening any oneline

This option is used to show the oneline diagram completely when opening any oneline. If the oneline was previously saved in a particular location in the diagram, if at the moment of opening a oneline this option is selected then the oneline will open showing the full oneline.

Edit Oneline Browsing Path

This option applies when you have [Online Links](#) included on a oneline diagram. Rather than specify the full path and name of a oneline diagram as a oneline link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the oneline file name stored with the link. This browsing path is also used when looking up [Shapefile Database R cords](#) and when finding the substation layout, URL, or command specified on the [Substation Information Dialog](#).

Save Onelines when Saving Case

By default, Simulator always saves any oneline diagrams that are open when the user chooses to save the case (pwb) file. This option allows you to choose to be prompted to save oneline diagrams when a case file is saved, or to never save onelines when the case file is saved.

Case Information Display Options

PowerWorld uses numerous [case information displays](#) to show [javascript:RelatedTopic1.Click\(\)](#) in tabular format. The options presented on this page of the PowerWorld Simulator Options Dialog control some of the general features of the case information displays.

Enterable Field Color

Fields whose values can be directly entered on the case information displays are colored navy blue by default. Click on the color field to change the color of enterable fields, or click the **Change** button.

Toggleable Field Color

Fields whose values can be toggled (changed) by left-clicking on them are colored green by default. Click on the color field to change the color of toggleable fields, or click the **Change** button.

At or Exceeding Limit Color

Fields whose values are at or exceeding a limit, are colored red by default. Click on the color field to change the color of such fields, or click the **Change** button.

Normal Field

Fields that cannot be modified directly from the case information display are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

Special External Field

Fields that can be modified through an auxiliary file or by pasting data into a sheet from a spreadsheet, but cannot be changed in a case information display, are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

Field not presently used

Fields whose values are overridden by other instances of the program are colored gray by default. Click on the color field to change the color of such fields, or click the **Change** button.

Background

Background of cells is colored white by default. Click on the color field to change the color of the background, or click the **Change** button.

Heading Background

Background of column and row headings is colored light gray by default. Click on the color field to change the color of the heading background, or click the **Change** button.

Data Fill Background Color

Background of selected cells when propagating values is colored yellow by default. Click on the color field to change the color of the heading background, or click the **Change** button.

Set Case Info Factory Default Colors

Clicking the **Dark Colors** button will reset the field and background colors to the defaults mentioned above, which comprise dark colors for fields, and light colors for backgrounds. Clicking the **Light Colors** button will set field and background colors to a specified set of light colors for fields and dark colors for backgrounds.

Highlight Objects with Selected? = YES

Check this option to automatically highlight all case information rows that represent a model object that has its "Selected" field set to YES. The row will then be highlighted using the color specified on this dialog. This option will apply to all case information displays representing all types of objects.

Save as Auxiliary File Data Format

This option allows the user to decide if the auxiliary files that can be saved from case information displays should be space delimited files or comma delimited files.

Key Fields to Use in Subdata Sections

This option allows the user to decide whether to use primary or secondary key fields or labels for the objects saved in the subdata section of an auxiliary file. In addition to being applied to the subdata section of auxiliary files, this option is also used in some special cases to dictate how elements are identified. The [Labels](#) topic contains additional information about this.

View/Modify Default Font

Clicking this button brings up a font dialog from which you can choose the font in which case information displays should show their data. Selecting a new font, font size, style, or color and pressing OK will change the default font, so that all case information displays will then employ a font having the selected properties. Not an individual case information display can override this setting by choosing a custom row height or font on the [Display Options tab of the Display/Column Options dialog](#).

Default Row Height

This option sets the default height of the rows in the case information displays. This field may need to be changed depending on the screen size and font size of the computer. By default, the height is set to 13. Not an individual case information display can override this setting by choosing a custom row height or font on the [Display Options tab of the Display/Column Options dialog](#).

Show Grid Lines

Grid lines are shown by default in case information displays. Unchecking this box will remove the grid lines from the displays.

Column Headings

This option allows you to choose whether the column headings of the case information displays are the normal column headings or are the variable names of the type of data stored in each column. Variable names are the strings used when writing data out to an [Auxiliary File](#). Change the headings to variable names can be helpful when looking at or creating auxiliary files.

Use Word Wrap

Check this box to wrap the heading column text, breaking it in several rows to fit the current column width. If unchecked, the column heading text will be displayed in only one row.

Highlight Key and Required Field Column Headings

If this option is checked, the key fields needed for identifying objects will be highlighted in the case information display. These fields are important if you are planning on pasting data back into Simulator from Excel or reading data in from an auxiliary file. If the necessary key fields are not present in the Excel paste of auxiliary file, the data in those formats cannot be processed by Simulator.

Similarly, the Required Field Column Headings can also be highlighted in the case information display. In order for Simulator to create new objects by reading them from an auxiliary file or and Excel paste, the required fields must be present in the pasted data or the object cannot be created.

Show Header Hints

This option allows the hints for the current field to be displayed when moving the cursor over the column heading.

Copy/Send Options

Typically when you copy information to the clipboard or send data to Excel from a case information display, the first two rows of the copied information contain the type of object the data represents (object name) and the column headings for each column of data. These rows are necessary if you intend to [paste the information back into Simulator](#), but they are unnecessary if you are only exporting data to another program with no intention of pasting the information back into Simulator. Thus, these two options allow you to choose which of the two rows, if either, you wish to have copied along with the actual data from a case information display when pasting in another application.

Disable Auto Refresh

This option prevents Simulator from automatically updating the contents of open case information displays with each solution. If this option is not checked, the data in all open case information displays will be updated automatically to reflect the system state calculated from each Power Flow Solution.

Set Factory Defaults

Clicking on this button will reset the options to their defaults.

File Management Options

There are two sub-categories on the File Management Options tab: PowerWorld Files, and EPC and RAW Files.

PowerWorld Files

Automatic Loading of Auxiliary File

A default auxiliary file can be loaded when the present case is opened by checking the **Automatically load an auxiliary file when the present case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with the present case. The full path needs to be included with the file name.

Automatic Loading of Auxiliary File with ANY Case

A default auxiliary file can be loaded when each case is opened by checking the **Automatically load an auxiliary file when ANY case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with each case. The full path needs to be included with the file name.

Automatic Archiving of PWB Files

This option allows you to effectively make backup copies of your working case every time you save the pwb file. For example, assume you have a case named Test Case.pwb. With the Automatic Archiving turned on, saving the case will first create a copy of the original file and rename it Test Case_1.pwb. The character used as a delimiter can be chosen optionally. The case with any changes you have just made will then be saved as the new Test Case.pwb file. Each time you save the case, the latest version is named Test Case.pwb, the last Test Case.pwb is renamed with the delimiter and 1 appended, and all other archived versions will be renamed with their number incremented by 1. The number of archive versions to maintain can also be chosen by setting the maximum number of archive files property. The automatically archived cases will be saved in the same directory from which the last case was opened.

Autosave Every ... Minutes

This option allows the user to specify how often (in minutes) the case is automatically saved. A value of zero means that this option will not be used.

Save Unlinked Elements of contingency, interface and injection group records in the PWB file

Checking this box allows saving any unlinked contingency, interface, and injection group records in the PWB case. The unlinked elements are only created when read from an auxiliary file containing unlinked records.

EPC and RAW Files

Display GE Additional Data in Case Information Displays

A large amount of data which is not relevant to the Simulator power flow solution is contained in the GE EPC file. This data is now all read and stored in Simulator and Simulator's PWB file format. By default columns which represent this extra data are not available when choosing [Display/Column Options](#) on a [Case Information Display](#). If you would like these fields to be available, check this box.

Load Additional Data from ECF

Older versions of Simulator did not read all the extra data from the GE EPC file and instead stored it in files with the ECF extension. If you would like to read in one of these old ECF files, click this button.

Remove Additional Data from Case

Because all of the extra data in the EPC can take up a lot of space (approximately doubles the PWB file size and increases the memory foot-print of Simulator itself too), you may want to just delete all this information. Click this button to permanently remove this data from your case.

GE EPC Motor Data Table

Part of the extra data read from the EPC file is Motor Data which does not map to an object used inside PowerWorld. To see this information, click on this button.

PTI RAW Transactions due to tie-line loads

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) are automatically created with the ID "RAW_LOAD". (You can optionally choose to load the RAW file "with options". This will allow you to choose whether or not the transactions should be added. See [Case Formats](#) for more information.) These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines only". If you know that your case was solved using "lines and loads", then click the button **Remove transaction due to tie-line loads**. Clicking this will delete all the MW Transactions with an ID of "RAW_LOAD". You may also click the button **Add transactions due to tie-line loads** to recreate a list of transaction which represent flows between loads with different areas than their terminal bus.

Transformer Regulated Bus Side

EPC and RAW files require the storage of which side of the transformer the regulated bus is on to determine which direction to move the tap in order to regulate voltage. In PowerWorld the tap sensitivity is always calculated when a transformer needs to move so this is not relevant. The buttons are provided to determine this side. If you have a solved power flow solution, then using tap sensitivities is good. If you do not have a solved power flow, then Recalculate using Closest Bus will also work (this just determines how far away the regulated bus is from the FROM and TO bus by summing up the total impedance that must be crossed to reach the regulated bus.)

hdbexport Files

Mapping of CBTyp to Branch Device Type

When loading an hdbexport CSV file, a Branch record in Simulator is created for each CB record. The Branch Device Type for these branches is defined based on the CB's CBTyp using the following rules:

if CBTyp contains "CB" --> Breaker

Else if contains "LBD" --> Load Breaker Disconnect

Else if contains "GND" --> Ground Disconnect

Else if contains "FUSE" --> Fuse

Else utilize user-Defined rules specified below

Else -- Disconnect

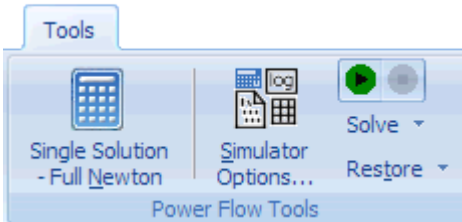
The **Defined rules specified below** can be defined in the bottom section of the hdbexport Files tab. The Priority can be set here for the CBTyp and the Branch Device Type can be specified with the following options: *Breaker, Disconnect, Fuse, Ground Disconnect, or Load Break Disconnect*.

Also after loading an hdbexport CSV file, if any CBTyp values were encountered that were not recognized a dialog box similar to the **Defined rules specified below** will appear prompting the user to designate the Branch Device Type rules for each CBTyp.

Power Flow Tools Ribbon Group

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Simulator Options

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using the either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

Play Button

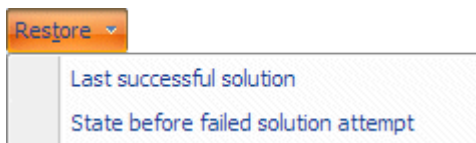
Click this button to cause repeated solutions of the system and to cause the animation of online diagrams

Pause/Stop Button

Click this button to stop the action of the Play button.

Restore Menu

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

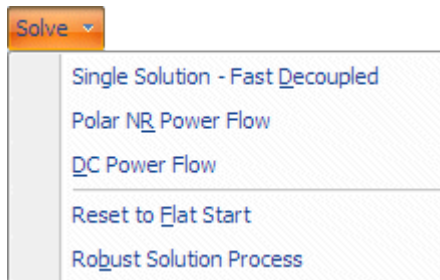
To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

Solve Menu



Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

Polar NR Power Flow

This performs a Newton Raphson power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

DC Power Flow

This automatically changes the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

Reset to Flat Start

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero.

Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

Robust Solution Process

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.

Solving the Power Flow

At its heart, Simulator is a Power Flow Solution engine. Power flow is a traditional power engineering calculation that is performed to determine the flows on all lines and the voltages at all buses in the system given the power injections at all buses and the voltage magnitudes at some of them.

The power flow problem entails solving a system of nonlinear equations. Solving a nonlinear system requires the use of an iterative algorithm to hone in on the correct solution. Many nonlinear system solvers have been developed, and PowerWorld provides access to the full Newton-Raphson method with an optimal multiplier and the fast decoupled method.

Usually, the power flow computation converges quickly. However, it is certainly possible to model conditions for which no Power Flow Solution exists, or for which the algorithm cannot converge to the solution within the [maximum number of iterations specified](#). For such situations, the [message log](#) will provide a message indicating that the computation failed to converge. Furthermore, unless [blackouts are disabled](#), the screen is grayed, and a message indicating a blackout has occurred is shown.

In order to calculate the power flow solution, simply press the Single Solution button on the main Program Toolbar. Clicking Single Solution actually performs several pre-processing activities and then runs three nested loops which solve the power flow: MW Control (Outer) Loop, Controller (Middle) Loop, and Power Flow (Inner) Loop.

Pre-Processing Activities

The Power Flow Loop is where the traditional power flow matrix equations are solved.

Angle Smoothing

When a transmission branch status is changed from OPEN to CLOSED across a branch that has a large voltage angle difference, this can introduce a very large initial power flow mismatch and cause the inner power flow loop to diverge. Angle Smoothing will alleviate this large angle difference by smoothing the angles in the system around the newly closed in branch resulting in much better power flow convergence. Angle smoothing will also work if a series of branches are all closed in together. A message will be written to the [message log](#) to indicate this is occurring. Angle smoothing should be enabled by default, but an option exists with the [Advanced Power Flow Solution Options](#) to disable this. Angle smoothing works best for individual branches or a series of branches that have been closed that are not electrically near other branches that have been closed. If modifying a case and adding in new transmission lines that are electrically near each other, it might be better for solution convergence to disable angle smoothing.

Generator Remote Regulation Viability

Generators are allowed to remotely regulate any bus in the system. However, if there is no transmission path between the generator terminal bus and the remotely regulated bus which does not pass over any other PV buses, then the generator will not be able to regulate this bus remotely. Additionally this will introduce a numerical condition that makes the inner power flow loop not converge. This pre-processing activity ensures catches this condition and prevents these generators from performing voltage regulation. A message will be written to the [message log](#) to indicate this is occurring.

Estimate MW Change Needed

When large changes in generation or load are made to the system, then ultimately this entire mismatch will show up at the island slack buses during the first inner power flow loop. This can cause the inner power flow loop to not converge. When automatic generation control is enabled for this system, then this pre-processing activity will automatically try to initialize generator outputs throughout the system to prevent the entire mismatch from appearing at the slack bus. Eventually the MW Control (Outer) Loop will be executed to bring generators to their proper outputs as specified by the Area or Island AGC choices.

Low Impedances Lines Voltage Profile

When pre-processing the voltage profile, Simulator will look at groupings of buses connected by very low impedances lines. If a bus in a grouping of energized buses has a zero voltage while other buses in the group do not, the zero voltage will be changed to the first non-zero voltage found in the grouping. This provides a much more reliable solution.

Estimate Voltages at Buses that Have Just Been Connected

As part of the pre-processing of voltages, voltages and angles at buses that have just had their status changed from *Disconnected* to *Connected* will be estimated assuming that the voltages and angles at buses that have not just changed status remain fixed. This will better facilitate power flow convergence. A message will be written to the [message log](#) to indicate this is occurring.

Three Nested Loops

Power Flow (Inner) Loop : Red Loop

The Power Flow Loop is where the traditional power flow matrix equations are solved. There are several [Common](#) and [Advanced](#) Solution options which affect this loop. In the [message log](#), a **RED** outline will be drawn around the inner power flow loop. Additionally, a **PURPLE** outline will be drawn around any generator Mvar limit checking which occurs inside this loop.

Controller (Middle) Loop : Green Loop

Once the Power Flow Loop is solved, control devices check if their control requirements are being met. Control devices are checked in the following order or precedence.

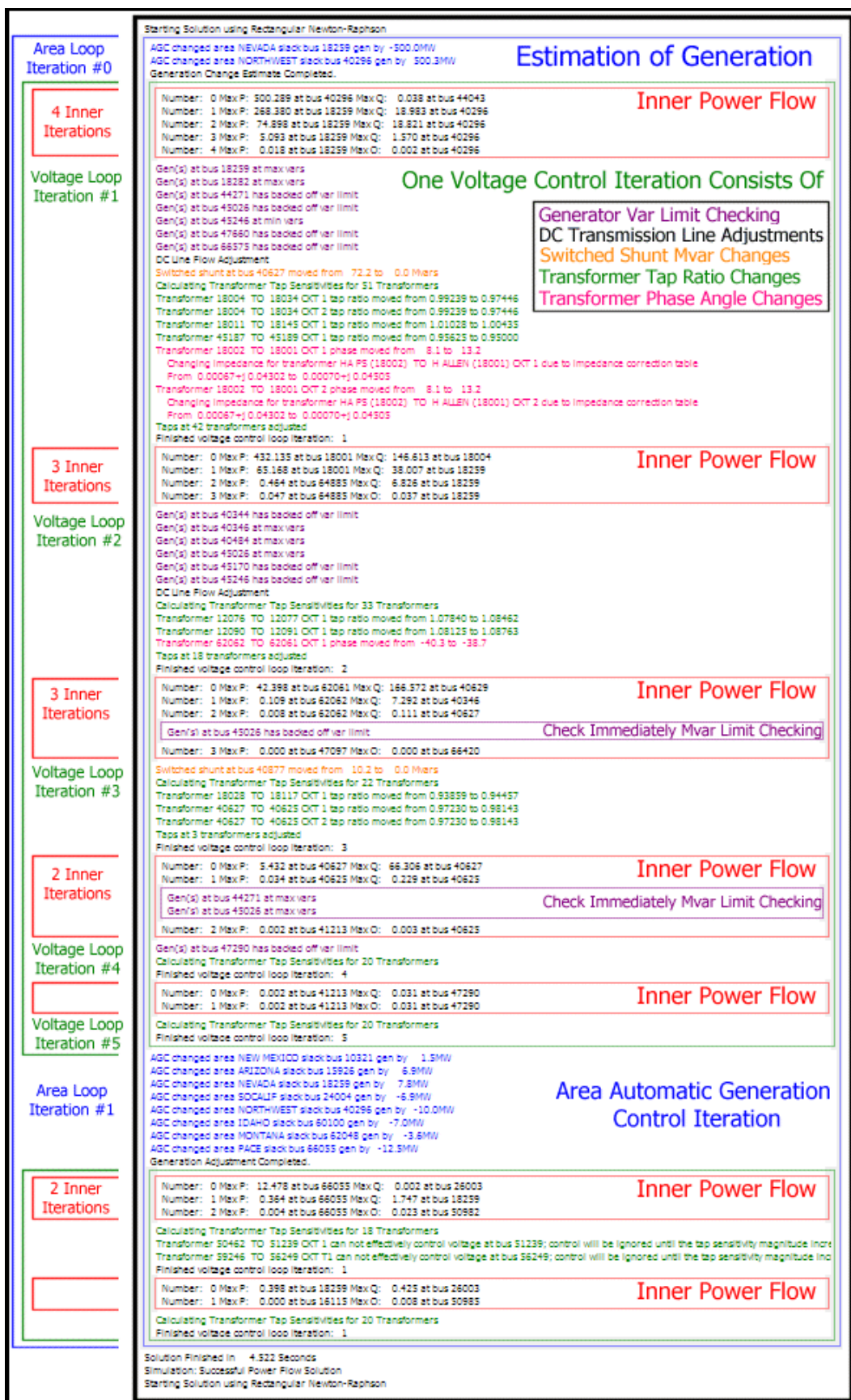
1. Generator Mvar Limit Checking
2. DC Line Solution
3. Switched Shunt Controls (each switched shunt or switched shunt control group is examined individually)
4. Load-Tap-Changing (LTC) transformers and Phase Shifter (can be coordinated switching or examined individually)

If any control devices requiring changes, then these changes are made and the power flow loop is re-solved. This continues until no more control loop changes are made. There are several [Common](#) and [Advanced](#) Solution options which affect this loop. In the [message log](#), a **GREEN** outline will be drawn around this loop.

MW Control (Outer) Loop : Blue Loop

After the Control Loop has completed, the MW Control loop is entered and generation (and possibly load) is moved to meet the MW control options set in the case. Normally, MW control is done by [area control](#) with each area varying generation to meet its own load, losses and interchange. However, you may also use [island-based control](#) to dispatch MWs by island. If any MW control is needed, then the Control Loop and Power Flow Loop interaction must be repeated and so on. In the [message log](#), a **BLUE** outline will be drawn around this loop.

A depiction of these loops and how you will see them represented in the message log is shown in the following figure.



Area Control

One of the most important aspects of interconnected power system operation is the requirement that each operating area changes its total generation to match changes in the sum of its load plus losses plus power transactions with other areas. This requirement is normally met by Automatic Generation Control (AGC). The purpose of AGC is to ensure that the actual MW output of an area is equal to the scheduled MW output of the area. The AGC system accomplishes this by first calculating the Area Control Error (ACE), which is defined as

$$ACE = P_{\text{actual}} - P_{\text{scheduled}}$$

In Simulator, $P_{\text{scheduled}}$ for an area is made up of the Area's [MW Transactions](#) and the Area's *Unspecified MW Export*. MW Transactions represent the transfer of power between two areas in the power system. This transaction is done presumably under a contract between the two areas. The advantage of using MW transactions to describe these is that it ensures that the total export of all areas is consistent: if one area is exporting 100 then another area is automatically importing that power. However, each area also specifies a value called the Unspecified MW Export which can be entered on the [Area Information Dialog](#). The unspecified MW export represents an export of power from the area that goes to an unspecified other area. When using unspecified MW exports, it is important that the total unspecified MW exports in the system sum to zero. PowerWorld highly recommends that care be taken when using unspecified exports.

Whenever the ACE is greater than zero, it means that the area is over generating and thus needs either to decrease generation or to sell more. Likewise, whenever the ACE is less than zero, the area is under generating and thus needs either to increase generation or to buy more. AGC works to keep the ACE close to zero.

In Simulator, there are six options for implementing AGC:

No area control

The output of the generators does not change automatically. You must manually change the generation to match system load/losses/transaction variation. All change in load/losses/transaction in this area will be made up at the islands slack bus.

Participation Factor Control

The output of all the area's generators who have their AGC field set to "YES" change automatically to drive the area control error (ACE) to zero. Each generator's output is changed in proportion to its participation factor. Checking this option enables the **Set Factors** button on the [Area MW Control Options](#) tab of the [Area Information Dialog](#), which, when pressed, opens the [Generator Participation Factors Dialog](#). Participation Factor Control only adjusts generation when a change to the system has taken place, such as changing the amount of load in the case, or defining new area to area transactions.

In Participation Factor Control, the ACE is allocated to each AGC generator in the area in proportion to that generator's participation factor divided by the total of the participation factors for all AGC generators in the area. A generator's participation factor cannot be negative. By default, a generator's participation factor equals its current MW setpoint value, but individual participation factors can be changed.

Economic Dispatch Control

The output of all AGC generators in the area changes automatically to drive the area control error (ACE) to zero. Each generator's output is changed so that the system is dispatched economically, based on cost information entered for the generators in the case. Note that cost data is not generally included in standard load flow data. Without realistic cost data entered into Simulator, the

use of the economic dispatch algorithm may not be very useful. Cost data must be obtained from another source and entered into a case in Simulator, either manually or through the use of [Simulator Auxiliary Files](#).

With Economic Dispatch (ED) Control, Simulator tries to change the output of the area's AGC generators economically so that the area's operating cost is minimized. ED control recognizes that some generators are less expensive than others and tries to use the least expensive generators to the largest extent possible.

To do economic dispatch, we need to know how much it would cost to generate one more MW at a particular generator. This is known as the incremental or marginal cost. For example, for the cubic cost curve model, the incremental cost for each generator is modeled using the formula:

$$\lambda_i = IC_i(P_{gi}) = (b_i + 2c_i P_{gi} + 3d_i (P_{gi})^2) * \text{fuelcost} \quad \$/\text{MWH}$$

The plot of $IC_i(P_{gi})$ as a function of P_{gi} is known as the incremental-cost curve. The economic dispatch for a system occurs when the incremental costs for all the generators (λ_i) are equal. This value is known as the system λ (lambda) or system incremental cost. Its value tells you how much it would cost to generate one more MW for one hour. The system lambda becomes important when trying to determine whether or not an area should buy or sell power. For example, if an area can buy power for cheaper than it can generate it, it might be a good idea for the area to buy power.

Generators that are allowed to participate in economic dispatch control will have their MW limits enforced regardless of how any of the options specifying that MW limits be enforced are set.

Area Slack Bus Control

Only the output of the area's slack bus changes automatically to drive the area control error (ACE) to zero. This type of generation control is usually only good for small disturbances to the injections and/or transactions in a case, and can often fail to find a solution when larger disturbances are examined.

Injection Group Area Slack Control

Only the Injection Group specified as the Injection Group Area Slack will change to automatically drive the area control error (ACE) to zero. This allows you to be very specific with each area as to which generation (or load) should vary to maintain ACE.

Optimal Power Flow (OPF)

The [OPF](#) option will only be available if you have the OPF add-on for PowerWorld Simulator. The OPF control is very similar to the Economic Dispatch control in that it attempts to dispatch generation to minimize costs. The additional function of the OPF is to minimize the costs while also obeying line, transformer, and interface limit constraints. This option is also not useful without realistic generator cost information, which usually must be obtained from another source and entered into Simulator to augment a load flow case.

The OPF control also relies on the cost curve in order to perform an economically optimal power flow. However, the OPF routine makes use of piecewise linear curves in its solution algorithm. This does not prevent you from entering the cost information as cubic cost models, described by the equation above. Rather Simulator's OPF routine allows you to specify how to break up the cubic curve and model it as a piecewise linear curve for the OPF algorithm.

Generators that are allowed to participate in OPF control will have their MW limits enforced regardless of how any of the options specifying that MW limits be enforced are set.

In addition, you can also enter piecewise linear curves directly instead of the cubic cost curve models. In fact, a mixture of piecewise linear and cubic models is acceptable. For the economic dispatch routine, whichever type of model is entered will be used directly for each generator. For the OPF

routine, all piecewise linear curves entered directly will be used as is, and any cubic models entered will be converted to piecewise linear curves internally during the processing of the OPF algorithm.

Area Control Error (ACE) Chart

The ACE chart plots the area control error for an area over time. For details on ACE, please see topic [Area Control](#). To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > ACE Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Load and Generation Chart

The Load and Generation chart plots an area's load + losses and generation in MW when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Load and Generation Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself and specifying the axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Losses Chart

The Area Losses chart plots an area's real power losses when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Losses Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area MW Transactions Chart

The Scheduled MW Transactions chart plots the scheduled real power (MW) transactions for an area when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area MW Transactions Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Area Average Cost Chart

The Average Cost per MWH chart plots the average cost per MWH for an area over time. This value is calculated by dividing the total cost of operating the area (generation cost + purchased power cost - revenue from power sales) by the MW load in the area. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Average Cost Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals. Note that as the area's load increases, the average cost per MWH tends to increase.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

Edit Mode Overview

The Edit Mode is used to create and/or modify cases and onelines. You can use the Edit Mode to create a case from scratch or to modify existing power flow cases stored in PowerWorld PWB or AUX files, in the PTI Raw data format, GE PSLF EPC data format, or the IEEE Common Format. New users may wish to view step-by-step tutorials on either

[Creating a New Case from Scratch](#)

[Creating Onelines for an Existing Power Flow Case.](#)

To enter the Edit Mode, select the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

A powerful capability of the Simulator is its ability to a case by graphically placing/editing on a oneline diagram. These display objects consist of both power system devices, such as buses, generators, and transmission lines; and that show various system parameters, provide descriptive text, or function as a static background.

Simulator's oneline diagrams illustrate the current state of the components of the power system. Most display objects correspond to records in the underlying power system model, but not all records in the power system model need to have an associated display object. In fact, for large system models, it may be that most of the system will not be illustrated. In such cases, you will want to devote more detail to the more critical areas of the system so as not to clutter the view. Furthermore, it is possible to associate more than one oneline with a single power flow case, and a single oneline may be associated with multiple cases. This great flexibility can prove to be a big time-saver. Please see [Relationship Between Display Objects and the Power System Model](#) for a more thorough discussion.

Edit Mode General Procedures

In order to simplify the process of graphically constructing a power flow case, Simulator's drawing interface obeys the following conventions for most objects:

Inserting a New Object

- Go to the [Draw](#) ribbon tab and use the menus on the [Individual Insert](#) ribbon group to add a single oneline object type you would like to add to the oneline.
- Left-click on the location on the display where you would like to position the object.
- Once the object is placed, Simulator displays a dialog box that allows you to specify various options for the object.
- If desired, use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the appearance of the object once it has been placed.

Moving, Resizing, and Rotating an Existing Object

- Select the object by clicking on the object with the left mouse button. Handles are displayed around the object to indicate it has been selected.
- To move the position of the object, place the mouse anywhere on the object except at a handle location. Then drag the object around the screen by holding the left mouse button down.
- To change the size of an object using the mouse, first select the object. Then place the mouse on one of the object's resizing (white) handles. The cursor will change to either a horizontal, vertical or diagonal two-headed arrow shape. Then drag the mouse to change the object's size. You can also specify the size of most objects using their dialog boxes.
- To rotate an object, first select the object. Then place the mouse over the rotation (green) handle in the upper left corner of the selected object. The cursor will change to a rotation symbol. Clicking and dragging will then rotate the object. Note that if an object (e.g., a pie chart or transmission line) does not have a rotation handle in the upper left, the object cannot be rotated.

Viewing/Modifying Object Parameters

- To view and/or change the options associated with a single object, right-click on the object. This either displays the object's dialog box directly, or it displays the object, from which you can elect to see the object's dialog box.

Selecting Several Objects to Modify Their Appearance

Hold down the Shift key while clicking objects on the screen to select several objects at once. You may then change the objects' attributes, such as , by using all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

To move the objects that are selected, click and hold down the left mouse button on any of the selected objects, drag the selected objects to a new location, and release the mouse button to place them. You can also move the selected objects by holding the SHIFT key and simultaneously using the UP, DOWN, LEFT, or RIGHT arrow keys (or use the Home, End, Page Up and Page Down keys to move objects).

To select a set of objects use the many options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects using a [rectangle, ellipse or polygon selector](#). You can also select only objects that meet some given criteria by using [Select by Criteria](#). As an example, you can

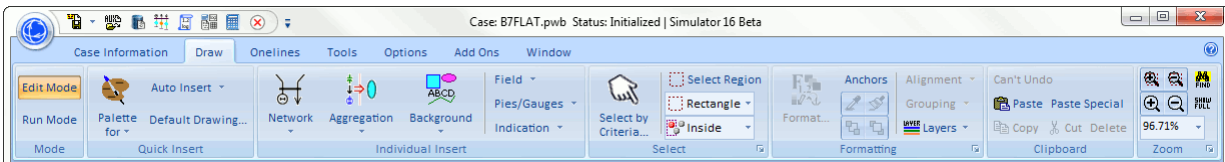
use Select By Criteria to select all the 345 kV transmission lines in a case. See [Select](#) ribbon group for more detailed help.

Changing An Object's Screen Appearance

Use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the screen appearance of either a selection of objects or the entire display.

Draw Ribbon Tab

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic.

Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.

Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

Online Display Options

The [Online Display Options Dialog](#) allows you to customize the appearance of the presently selected online diagram. To view this dialog, either select **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab, or choose **Online Display Options** from the online's [local menu](#). Please see the [Online Display Options Dialog](#) help for more information.

Anchored Objects

While in [Edit Mode](#), Simulator allows certain objects to be attached, or *anchored*, to another object, called the *anchor*. When an object that functions as an anchor is moved, all objects that are anchored to it will move with it. This feature can be very useful when you move objects around the oneline diagram in Edit Mode.

Anchoring has the property of "stacking" in Simulator. In other words, one object is anchored to another, which is in turn anchored to yet another. The best way to describe this is by example. A generator text field can be anchored to a generator. The generator, in turn, can be anchored to its terminal bus. If you move the terminal bus, both the generator and its anchored fields also move with the bus. However, if you just move the generator itself, only the generator fields will move with it. The bus and all other objects anchored directly to the bus remain in their original location.

If anchors are deleted it is generally necessary to reset anchors on anchored objects go to the [Draw](#) ribbon tab and choose **Refresh Anchors** on the [Formatting](#) ribbon group.

There are several types of anchored objects:

[Buses](#)

[Loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), [transmission lines](#) and [voltage gauges](#) may be anchored to their associated bus. When the anchor bus is moved, these anchored objects will move with it.

[Substations](#)

Similar to buses, any [loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), and [transmission lines](#) may be anchored to their associated substations. When the anchor substation is moved, these anchored objects will move with it.

[Generators](#)

[Generator fields](#) may be anchored to their associated generator. When the anchor generator is moved, these anchored fields will move with it.

[Loads](#)

[Load fields](#) may be anchored to their associated load. When the anchor load is moved, these anchored fields will move with it.

[Switched Shunts](#)

[Switched shunt fields](#) may be anchored to their associated switched shunt. When the anchor switched shunt is moved, these anchored fields will move with it.

[Area/Zone/Super Area Objects](#)

[Interfaces](#) can be anchored to area/zone/super area objects.

[Lines and Transformers](#)

[Circuit breakers](#), [line flow pie charts](#), and [line fields](#) may be anchored to their associated line/transformer. When the line/transformer is moved, these anchored objects will move with it.

[Interfaces](#)

[Interface fields](#) and [interface pie charts](#) can be anchored to their associated interface.

Inserting and Placing Multiple Display Fields

Simulator provides a convenient method of adding display fields to a variety of oneline display objects and placing them in default positions relative to the display object. Unlike the field placement implemented by selecting **Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, the method described here allows you to place multiple fields around a display object in a single operation. This option is available for several different types of objects, including buses, generators, lines and transformers, shunts, and loads. The option can always be accessed by right clicking the display object of interest and selecting the **Add New Fields Around ...** option from the resulting local menu. This brings up the **Insert New Fields around Selected Objects Dialog**. This dialog is divided into several tabs by object type. Only tabs that correspond to the type of objects you have selected will be available.

The tabs for inserting and placing multiple display fields for the various display objects (buses, generators, lines/transformers, shunts, and loads) are virtually identical in content. The tab illustrates the possible locations of the display fields for the various orientations of the display object. For example, since buses may be oriented either horizontally or vertically, the dialog shows how each of the eight possible bus fields would be positioned for each of the two orientations. Generators, loads, and shunts each have four possible orientations, so the dialog identifies the locations for the possible fields for each of the four orientations. Transmission lines and transformers can assume only one orientation, and the dialog will thus show the possible field locations for that single orientation.

Each field location is identified on the illustrations with a label of the form Pos #. In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), or the [Line Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**.

Click **OK** to implement your choices for field additions and placement, or click **Cancel** to discard the changes.

This help topic has gone over how to add new fields to existing display objects. It should be noted from this discussion that all objects have an associated set of default fields that will be added to the oneline when the objects are originally inserted. You may redefine the default fields selecting **Default Drawing Values** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab and modifying the [Default Drawing Options Dialog](#).

Setting Default Drawing Options

The Default Drawing Options Dialog is used to define the default sizes of new display objects, as well as various other display parameters. These options are available only in Edit Mode.

To open this dialog choose **Default Drawing Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Online](#) ribbon tab.

The Default Drawing Values Dialog is organized using a list showing the types of objects that can be added to onelines. To view information about a particular type of object, select that object type from the list and tables which describe the defaults available for that object will be populated. Two tabs will be available. One will contain the majority of default options that are available for the selected display object type, and the other will contain the default positions of fields that will be inserted when the object is inserted.

Generally, the Default Drawing Values apply when you insert *new* objects on an oneline diagram. Changing the Default Drawing Values does not automatically affect the *existing* objects already drawn on the diagram. However, you can select multiple objects on a oneline diagram, and right-click on any one object to invoke the object's popup menu. You will find the option **Apply Default Draw Values**, which will modify the formatting of the selected objects to meet the default drawing value specifications.

Variable Defaults depending on Voltage Level

Many kinds of objects can have more than one set of defaults specified depending on their voltage level. When more than one set of defaults is specified, the table is always sorted by Nom kV.

A new object is inserted with properties corresponding to the default with the lowest Nom kV that the object's nominal kV is larger than. If the object's nominal kV is smaller than all defaults, then it will be set according to the default with the lowest nominal kV.

For instance, assume 4 sets of defaults are defined for buses roughly as follows

Nom kV Size

>400 15

>300 10

>200 8

>100 5

A new bus with nominal voltage of 345 kV is inserted with size 10.

A new bus with nominal voltage of 299 kV is inserted with size 8.

A new bus with nominal voltage of 69 kV is inserted with size 5.

In order to add another set of defaults for a kind of object, right-click on the table and choose **Insert** from the local menu. In order to delete a set of defaults, right-click on the table and choose **Delete**. Once you have inserted a new default, specify the new Nom kV for the default along with the new defaults.

Notes: When you change the Nom kV of a default, the list always sorts itself according to Nom kV. **Insert** will not be enabled in the local menu for objects that cannot have more than one set of defaults. **Delete** will not be enabled if only one set of defaults is defined.

Modifying Values in the Default Drawing Values Dialog

The font colors in tables of this dialog follow the conventions of the [Case Information Displays](#). Most default values are enterable and can be modified by selecting the value you want to change and

then typing in the new value. Other values are toggleable and can be changed by double-clicking on the value. For reference regarding what the various "size" defaults mean, when the zoom level is at its nominal value (100%), the size of the screen is 100 by 100. There are also some special kinds of fields in this dialog which are discussed next.

Editing Colors and KV-Based Color Coding

In order to modify a color value, double-click on the colored rectangle to bring up the Color Dialog. Then specify the color you want. For some defaults, such as fill color, it is possible to specify the default color to be *none*. In order to set a color default to *none*, select the colored rectangle and press the Delete key.

At times, a user may want to see the kV level of screen objects directly from the oneline diagram, without having to open a text display. One way to do this is to color code the elements on the oneline diagram according to kV level. Right-click on the table and choose **Insert** from the local menu. Specify the Nom kV and color.

Editing Default Field Positions ("Pos1", "Pos2", etc...)

Many objects allow you to specify some default fields to be automatically added when a new object is drawn. For example, you may want to insert the bus name next to all new Bus Objects. For objects that allow the insertion of fields, a tab with field position diagrams will be available when that object type is selected. In order to specify the default positions of fields, click on the tab showing the field positions and click on the table row corresponding to the Nom kV of the defaults of interest. For object types that do not allow specifying the defaults by Nom kV, simply update the given position diagram. The position diagram will show the present settings for the new fields. Positions that have a default specified will be highlighted and the name of the default field will be shown. Positions with no default field will not be highlighted and will say "Pos1", "Pos2", etc.... For objects that have more than one possible orientation (e.g. generators can be up, right, left or down), there are multiple position diagrams showing the positions for each orientation.

In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), [Line Field Options](#), [Interface Field Options](#), [Substation Field Options](#), [Area Field Options](#), [Super Area Field Options](#), or the [Zone Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**. You can also modify default fields by double-clicking on the table for "Pos1", "Pos2", etc... This brings up the appropriate Field Options Dialog as well. If you press the Delete key while on a "Pos" field in the table, it will set the default field to "none".

Editing Stub Size and Stub Space

When utilizing the [Automatically Insert Transmission Lines](#) feature, the **Stub Size** and **Stub Space** values are used. Simulator will draw each automatically inserted branch such that it emerges from both its terminal buses at right angles. To accomplish this, Simulator draws each automatically inserted branch in three segments: two stubs perpendicular to the terminal buses having a length specified by the value supplied for **Stub Size**, and a third segment joining the two stubs. The amount of space between lines as they converge into a bus is set by **Stub Space**.

If you do not want transmission line stubs to be inserted, then set **Stub Size** to *none* or to a negative number. Note in order to set a value for **Stub Space**, **Stub Size** must be set to a positive number first.

Editing CB Size

When you insert new transmission lines or transformers, circuit breakers will automatically be inserted on the branch with their size specified by CB Size. If you do not want Circuit Breakers inserted for a particular voltage level, then set CB Size to *none* or a negative number.

Editing Pie Size

When you insert new transmission lines or transformers, pie charts will automatically be inserted on the branch with their size specified by Pie Size. If you do not want pie charts inserted for a particular voltage level, then set Pie Size to 'none' or a negative number.

Set Default Font

Click on this button to set the default font used for new text fields. This is the font which will be used for all new text fields. Note however that the default font size set using this dialog is not used *unless* the option **Use the default font size for new text objects** is selected. Otherwise, the specific kind of object and voltage level specifies what the font size should be. For example, in the Interface tab, one of the defaults is Font Size. New Interface Fields will use this font size.

Only Cut/Copy Display Objects, Not Power System Records

When you cut or delete an object from the online, Simulator needs to know whether you simply want to delete the display object from the online or to purge the definition of the power system object from the model. Check this box to tell Simulator to assume that it should always delete objects just from the online display, not from the power system model.

Online / Bus View Background Color

To change the default online background color, click on the Change button to select a new color. This color applies to both new online diagrams and the background color of the [bus view online](#) diagram.

Recommended Multi-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. Multiple sets of default options will be specified based on voltage level for objects for which it is beneficial to do this (i.e. transmission lines).

Recommended Single-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. A single set of default options will be specified for each object type.

Save All to Aux File and Load All From Aux File

The Default Drawing Options may also be stored in an Auxiliary File by Clicking the Save All to Aux File button to save all these customizations to an Auxiliary File or by clicking the Load All From Aux File to load all the custom hints from an Auxiliary File.

Area Display Objects

Area records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by area.

To insert an Area object, select **Aggregation > Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Area object should be placed. This will open the [Area Display Options](#) dialog, which will allow you to choose the area information to display, and set parameters for the displayed object. It is important to note that area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new area record to the load flow case, you need to change the area designation of a device in the load flow case to a new area number, or you need to insert a new area from the [Area Records](#).

Area objects can also be added by using the [Area insert palette](#).

Area Display Options Dialog

When you insert an area display object on a oneline diagram, Simulator opens the Area Display Options dialog. This dialog is used to control various display and identity attributes of the area display object. The dialog contains the following fields:

Number

This dropdown box lists the number of all areas in the case. Use this control to associate the display object with the correct area.

Name

If you would prefer to search through areas by name rather than by number, use the *Name* dropdown box to see a list of names of all the areas in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Area** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Area Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Zone Display Objects

Zone records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by zone.

To insert a Zone object, select **Aggregation > Zone** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Zone object should be placed. This will open the [Zone Display Options](#) dialog, which will allow you to choose the zone information to display, and set parameters for the displayed object. It is important to note that zone records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new zone record to the load flow case, you need to change the zone designation of a device in the load flow case to a new zone number, or you need to insert a new zone from the [Zone Records](#).

Zone Display Options Dialog

When you insert a zone display object on a oneline diagram, Simulator opens the Zone Display Options dialog. This dialog is used to control various display and identity attributes of the zone display object. The dialog contains the following fields:

Number

This drop down box lists the number of all zones in the case. Use this control to associate the display object with the correct zone.

Name

If you would prefer to search through zones by name rather than by number, use the *Name* drop down box to see a list of names of all the zones in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Zone** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Zone Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or as an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Super Area Display Objects

Super area records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by super areas.

To insert a Super Area object, select **Aggregation > Super Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Super Area object should be placed. This will open the [Super Area Display Options](#) dialog, which will allow you to choose the super area information to display, and set parameters for the displayed object. It is important to note that super area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new super area record to the load flow case, you need to insert a new super area from the [Super Area Records](#).

Super Area Display Options Dialog

When you insert a super area display object on a oneline diagram, Simulator opens the Super Area Display Options dialog. This dialog is used to control various display and identity attributes of the super area display object. The dialog contains the following fields:

Number

This field is unused for super area objects. Super areas are identified by a unique super area name instead.

Name

Use the *Name* dropdown box to see a list of names of all the super areas in the case.

Show Record Type Prefix

Check this box if you wish to place the prefix **Super Area** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Super Area Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle, a rounded rectangle, or an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Owner Display Objects

Owner records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include summary objects by owner, in which information about the generation and load of the owner are indicated with the object.

To insert an Owner object, select **Aggregation > Owner** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the owner object should be placed. This will open the [Owner Display Options](#) dialog, which will allow you to choose the owner information to display, and set parameters for the displayed object.

Owner Display Options Dialog

When you insert an owner display object on a oneline diagram, Simulator opens the Owner Display Options dialog. This dialog is used to control various display and identity attributes of the owner display object. The dialog contains the following fields:

Number

The owner number.

Name

The owner name.

Show Record Type Prefix

Check this box if you wish to place the prefix **Owner** before the name/number caption in the object.

Prefix Text

Specify additional prefix text to be added before the **Owner** prefix.

The remainder of the choices presented on the *Owner Display Options* dialog pertain to the object's display appearance.

Style

Choose whether the display object should appear as a rectangle or as an ellipse.

Caption

Indicate how the display object should be identified to the user: by name, number, or both.

Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

Area Fields on Onelines

Area fields are used to show various values associated with a particular area of the power system. Right clicking on the area field displays the [Area Field Dialog](#).

Edit Mode

To enter a new area field, first select **Field > Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click on or near the area object or a bus in the area for which you want to add a field. This calls up the [Area Field Dialog](#). Verify that the area number is correct. By default, this value is the number of the area associated with the closest area object, or if there are no area objects, the closest bus. Enter the total number of digits the field should display as well as the number of digits to the right of the decimal point. Depending on what the field is designed to display, you may need to enter an additional area number. Finally, select the field type. Click **OK** to save the field or **Cancel** to abort the operation.

With most types of area fields, an **Area Number** of 0 is valid and defines the field as showing values for the entire system.

To modify the parameters of an existing area field, position the cursor anywhere on the area field and right-click. This again brings up the [Area Field Dialog](#). Use the [Formatting](#) options to change various display attributes for the area field, including its font and background color.

Zone Fields on Onelines

Zone field objects are used to show different values associated with zones and the system. This dialog is used to view (and in a few cases modify) the parameters associated with these zone fields. To reach this dialog, select **Field > Zone Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and click the desired field location on a online diagram in Edit Mode, or right-click on an existing zone field on a online diagram.

Zone Number

Zone number associated with the field. When you insert fields graphically, this field is automatically set to the zone number associated with the closest bus on the online. With most types of zone fields, a *Zone Number* of 0 is valid and defines the field as showing values for the entire system.

Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

Total Digits in Fields

Total number of digits to show in numerical fields.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point in numerical fields.

Rotation Angle in Degrees

The angle at which the text is to appear on the online diagram.

Other Zone Number

Some of the fields, such as **MW Flow to Other Zone**, require that a second area be specified. If applicable, enter the second (other) zone here.

Delta per Mouse Click

This value is used only with the **Sched Flow to Other Zone** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Zone**, a spin button is shown to the right of the zone field. When the up spin button is clicked, the flow to the other zone is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

Field Value

Shows the current output for the zone field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Zone** field type only, you can specify a new value in MW. Exports are assumed to be positive.

Field Prefix

An optional string that precedes the field value.

Anchored

If the field is associated with a Zone Object on the diagram, the field can be anchored to the object so that if the object gets moved on the diagram, the field will move with it.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of zone field to show.

Name

Zone name.

Number

Zone number (1 - 999).

MW Load, Mvar Load

If the zone number is nonzero, then these fields show Total MW or Mvar load for the zone. If the zone number is zero, these fields show the total load in the entire system.

MW Generation, Mvar Generation

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar generation for the zone. If the *Zone Number* is zero, these fields show the total generation in the entire system.

MW Shunts, MVR Shunts

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar shunt compensation for the zone. If the *Zone Number* is zero, these fields show the total shunt compensation in the entire system.

MW Flow to Other Zone, Mvar Flow to Other Zone

Total MW or Mvar flow from the zone specified in the *Zone Number* field to the zone specified in the *Other Zone Number* field. The Zone Number field must correspond to a valid zone. If the *Other Zone Number* field is zero, this field shows the zone's total MW or Mvar exports.

MW Losses, MVAR Losses

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar losses for the zone. If the *Zone Number* is zero, these fields show the total real or reactive losses in the entire system.

Load Schedule Multiplier

Indicates the current value of the MW multiplier applied to the zone's loads.

Select a Field

Allows you to choose any other field associated with a zone not explicitly listed as an option in the Type of Field selection.

Select *OK* to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

Super Area Fields on Onelines

To display certain information about a super area, such as MW Load or MVAR losses, insert a super area field. This can be done in Edit Mode by selecting **Field > Super Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Super Area Field Information](#). From here you can choose which super area to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 12 different field options to choose from. If a field value is not defined, question marks will be shown.

Owner Fields on Onelines

Owner field objects are used to show different values associated with owners. This dialog is used to view and modify the parameters associated with these fields. To insert an Owner field, click on **Field > Owner Field** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Owner Number

Owner number associated with the field. When you insert fields graphically, this field is automatically set to the owner number associated with the closest bus on the oneline.

Find by Number

To switch to a different owner in the field options dialog, you can enter the number in the **Owner Number** field, and press the **Find by Number** button to update the dialog with information for the new owner.

Owner Name

Name of the owner whose information is presently being displayed in the dialog.

Find by Name

To switch to a different owner in the field options dialog, you can enter the name in the **Owner Name** field, and press the **Find by Name** button to update the dialog with information for the new owner.

Total Digits in Fields

Total number of digits to show in the field.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Prefix

An optional string that precedes the field value.

Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

Field Value

Shows the current output for the owner field. Whenever you change the **Type of Field** selection, this field is updated.

Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

Type of Field

Used to determine the type of owner field to show. You can choose from one of the six default fields, or click on the **Find Field...** button to choose from a list of all available owner fields.

Bus Display Objects

In power system analysis, the term "bus" is used to refer to the point where a number of electrical devices, such as lines, loads or generators, join together. On the oneline diagram, buses are usually represented with either a thick horizontal line or a thick vertical line. The bus thickness and color can be customized.

Right-clicking on the bus will display its local menu. The local menu offers you the chance to view the corresponding [Bus Dialog](#), the [Quick Power Flow List](#), and the [Bus View Display](#). When the application is in Edit Mode, the local menu will also allow you to add bus fields to the bus and to insert any undrawn buses connected to the selected bus. [Bus Fields](#) are often placed close to the bus to indicate its voltage magnitude, voltage angle, and other relevant information.

Edit Mode

To add a new bus to the case graphically, follow this simple procedure:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group of the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the oneline background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#).
- Use the Bus Option Dialog to specify the number, name, size, thickness, orientation, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to it. Every bus must have a unique number.
- The Bus Option Dialog will include a message just above the Bus Number field indicating that a new bus will be inserted into the power system data model. This will appear as long as the Bus Number that is selected does not already exist in the power system case.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. If you do not wish to add the bus to the case, click **Cancel**.

If you are simply adding a symbol to the oneline diagram for a bus that has already been defined in the case, many of the parameters you are asked to specify in step three will be filled in for you.

To modify the parameters for an existing bus, position the cursor on the bus and right-click to invoke the bus' local menu. From the local menu, choose **Bus Information Dialog** to view the associated [Bus Dialog](#). You may change any of the parameters specified there. When a bus' number is changed, the bus numbers associated with all of the devices attached to that bus are also automatically changed. To renumber several buses simultaneously, please see [Bus Renumbering Dialog](#).

To modify any aspect of a bus' appearance, first select the bus, and then click any of the format buttons found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. You can change the length of the bus (but not its thickness) by dragging the bus' resizing handles.

In order to delete an existing bus after selecting it, use either the **Cut** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to preserve a copy of the bus on the Windows clipboard, or **Delete** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to remove the bus without copying it to the clipboard. You will be asked whether you want to remove both the display object and its associated bus record, or merely the display object, leaving the bus in the power flow model. If you will never be deleting a record from the power system model, you may also choose the option labeled **Always Delete Objects Only**. Be careful when deleting existing buses with attached devices. An error will occur during validation if you do not also delete the attached devices or attach them to other buses.

Bus Fields on Onelines

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to easily change bus device properties.

Run Mode

Right clicking on a bus field gives you the option to open the [Bus Field Dialog](#) or the [Bus Information Dialog](#).

Edit Mode

Simulator offers two options for adding bus fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the bus to which you want to add the field. This invokes the [Bus Field Dialog](#). Enter the bus number associated with the device (the default is the closest bus to the field), the total number of digits to show, and the number of digits to the right of the decimal point. An optional Field Prefix can be used. Choosing to Include Suffix will include relevant units for the type of field selected. The field can be Anchored to the bus that it represents and the field can be rotated by a specified Rotation Angle in Degree. Finally, select the type of field to show.

The second approach for adding new bus fields entails right-clicking the bus and selecting **Add New Fields Around Bus** from the resulting local menu. See [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing bus field, position the cursor anywhere on the object and right-click. This brings up the [Bus Field Dialog](#). Choose any of the format buttons on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

Old Voltage Gauges

Voltage gauges provide a way to visualize the voltage of a bus relative to its high- and low-voltage alarm limits. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Voltage gauges in Simulator work the same way. A voltage gauge has three markings on its side, each of which identifies a key per-unit voltage level. Specifically, these three markings locate the minimum, maximum, and target per-unit voltages. Often, the target voltage level is the nominal voltage, but this is not a requirement. Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the voltage gauge, you associate it with a bus, and you specify its fill color and its minimum, maximum, and target voltage levels. Once the gauge has been placed on a display, it will reveal changes in its associated bus's voltage by varying the height of its filled region. This tool was introduced to provide an alternative to voltage contours to show the variation of voltage across a region. [Contouring](#) can reveal the variation of only a single quantity at a time. For example, it is impossible to contour bus voltage magnitude and bus voltage phase angle simultaneously. Voltage gauges are helpful because they allow you to show the voltage profile superimposed on a contour of some other quantity.

To add a voltage gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Bus** from the **Individual Insert Ribbon Group** on the [Draw](#) ribbon tab. The cursor will become a crosshair. Click on the oneline diagram where you would like the new voltage gauge to appear. The [Voltage Gauge Options Dialog](#) will appear. Use this dialog to define the minimum, maximum and target voltages for the voltage gauge, as well as its fill color and whether it should be anchored to its associated bus. After you click the "OK" button, the Voltage Gauge Options Dialog will close, and the new voltage gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the voltage of its associated bus changes. To modify any of the characteristics of the gauge, such as its key voltage levels, fill color, and anchor setting, simply right-click on the bus voltage gauge to open the Voltage Gauge Options Dialog again.

Old Voltage Gauge Options Dialog

The Voltage Gauge Options Dialog is used to define and configure a [bus voltage gauge](#). A bus voltage gauge is associated with a particular bus and reveals the bus's voltage relative to specified minimum, maximum, and target per-unit voltage levels. The height of the colored column in the voltage gauge indicates the bus's voltage relative to these markings.

The dialog has the following controls:

Number, Name, and Find

Use the **Number** and **Name** dropdown boxes to identify the bus to which you want the gauge to correspond. Select a bus number from the Number dropdown box to identify the bus by number, and a bus name from the Name dropdown box to identify a bus by name. It may be more convenient to press the **Find** to open the [Find Dialog](#), which allows you to specify a bus by either name or number using wildcards. When you first open the Bus Voltage Gauge Dialog, the bus name and number will correspond to the bus object that was closest to the point where you clicked.

Minimum, Target, and Maximum

Use these three spin edit boxes in the **Voltage levels** group box to specify the minimum, maximum, and target per-unit voltage levels. These settings determine where on the gauge its three markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current bus, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

Fill color

The Fill color box reveals the color that will be used to paint the filled region of the gauge. Click on the Fill color box to open a Color Dialog, which you may then use to specify a different color.

Anchored

A bus voltage gauge is said to be [anchored](#) if, when you move its associated bus object, it moves with it. Check the Anchored check box to ensure that the gauge will move with its associated bus. Otherwise, when you move its associated bus object, the gauge will stay in its current position.

OK, Help, and Cancel

Click OK to finalize your settings. This will create a new voltage gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes. Click the Help button to reveal this help text.

Substation Display Objects

Substations in Simulator define a group of buses that are closely connected. Each bus can belong to either one substation or no substation (called unassigned). By default in Simulator, all buses are not assigned to a substation. This is done because all traditional text file power flow formats do not include information regarding a bus' substation.

Substations are represented on the oneline as a rectangle with the name of the substation inside it. Other information about the substation is also displayed on the rectangle:

- The upper left corner displays a generator symbol if generation exists in the substation.
- The upper right corner displays a load symbol if load exists in the substation.
- The lower right corner displays a shunt symbol if shunts exist in the substation.
- The lower left corner displays the number of buses inside the substation.
- The lower middle displays the maximum voltage level in the substation.

You can also customize the size, colors, and font name and style of a substation object. Note however that the font size of the substation object is automatically changed by Simulator as you change the size of the rectangle.

Right-clicking on the substation will display its local menu. The local menu offers you the chance to view the corresponding [Substation Information Dialog](#) and the [Substation View Display](#).

Substation Fields on Onelines

To display certain information about a substation, such as MW Load or MVAR losses, insert a substation field. This can be done in Edit Mode by selecting **Fields > Substation Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Substation Field Options](#) dialog. From here you can choose which substation to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 8 different field options to choose from. If a field value is not defined, question marks will be shown.

Generator Display Objects

Generators are represented on the oneline as circles with a rotor symbol inside. The default rotor symbol is a "dog bone," but other symbols can be selected. Multiple generators at a bus are allowed, with each being distinguished by a unique character identifier.

Each generator symbol (except that corresponding to the slack) can be equipped with a circuit breaker (see that can be used to change the status of the generator. You may toggle the generator status by clicking on the circuit breaker while in Run Mode.

[Generator fields](#) are often placed close to the generator on the oneline to indicate the generator's MW/Mvar output or other information associated with the generator.

Run Mode

When [animation](#) is active, the default flow of the arrows emerging from the generator is proportional to its MW output. You can customize the appearance of this flow using the Animated Flows Tab of the [Oneline Display Options Dialog](#).

Right clicking on the generator brings up the generator submenu. This menu is used to display a variety of information about the generator, including

- [Generator Information Dialog](#)
- [Input-output curve](#)
- [Fuel cost curve](#)
- [Incremental cost curve](#)
- [Heat-rate curve](#)
- [Reactive capability curve](#)

Edit Mode

To add a new generator to the case, select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, place the cursor on the bus where you would like to attach the generator, and click with the left mouse button. This calls up the [Generator Dialog](#). The bus number is automatically determined from the bus to which you attached the generator. The ID field contains a alphanumeric ID used to distinguish multiple generators at a bus. The default is '1'.

Enter the size, the thickness of the lines (in pixels) used to display the device, orientation, rotor symbol, and other parameters for the generator. Each generator can optionally contain a circuit breaker for connecting or disconnecting the device in Simulator. Use options found with the [Display Object Options](#) on the [Oneline Display Options](#) dialog to specify if a circuit breaker should be shown. Select OK to add the generator. If you do not want to add the generator to the case, select Cancel.

To modify the parameters for an existing generator, position the cursor on the generator and right-click. This again brings up the Generator Dialog. You can then change any parameter (be careful in renumbering an existing generator). Use the [Formatting](#) ribbon group commands to change the color, line thickness, and other display parameters.

Generator Fields on Onelines

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

Run Mode



For generator fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Generator Field Dialog](#) or the [Generator Information Dialog](#).

Edit Mode

Simulator offers two options for adding generator fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Generator Field** from the **Individual Insert** ribbon group on the [Draw](#) ribbon tab and click near generator for which you want to add the field. This invokes the [Generator Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest generator to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show. For generator actual MW and Setpoint MW types and load MW and Mvar fields, specify a nonzero value in the *Delta per Mouse Click* to design a bus field with an integrated spin control. The Gen AGC Status field is used to display the automatic generation control status of the generator. The user can toggle this status in Simulator by clicking on the field. Likewise, the Gen AVR Status field is used to display the automatic voltage regulation status of the generator. Again, the user can toggle this status by clicking on the field.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Generator* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing generator field, position the cursor anywhere on the object and right-click. This brings up the [Generator Field Dialog](#). Choose , or  from the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

Load Display Objects

Simulator models aggregate load at each system bus. Multiple loads at a bus are allowed. Each load object on the oneline comes equipped with a circuit breaker. The status of the load corresponds to the status of its circuit breaker. A circuit breaker is closed if it appears as a filled red square, and it is open if it appears as a green square outline. In Run Mode, you may toggle the status of the load by clicking on its associated circuit breaker.

[Load fields](#) are often placed close to the loads on the oneline to indicate their MW/Mvar value.

Run Mode

When [animation](#) is active, the flow of the arrows into the load is proportional to its current MW load. You can customize the appearance of this flow using the Animated Flows Tab of the [Online Display Options Dialog](#).


Right clicking on a load (bus) field gives you the option to open the [Load Field Dialog](#) or the [Load Dialog](#).

Edit Mode

To add a new load to the case, select **Network > Load** from [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, select the bus to which you want to attach the load with the left mouse button. This calls up the [Load Dialog](#). The bus number is automatically determined from the bus to which you attached the load. The ID field contains a two-character ID used to distinguish multiple loads at a bus. The default ID is 1.

Enter the size, the thickness of the lines [in pixels] used to display the device, the orientation, and the base MW and Mvar load values for the device. Usually, only the Constant Power fields are specified as nonzero. The Constant Current and Constant Impedance fields are used to specify loads that vary with voltage. Constant current loads vary proportionally with bus voltage, while constant impedance loads vary with the square of the voltage. Specify the constant current and constant impedance values assuming one per-unit voltage.

Select OK to add the load. If you do not want to add the load to the case, select Cancel.

To modify the parameters for an existing load, position the cursor on the load and right-click. Select *Load Information Dialog* from the local menu to invoke the [Load Dialog](#). You can then change any parameter as desired. You can select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the drawing parameters of the load.

Load Fields on Onelines

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

Run Mode



For load fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a load field gives you the option to open the [Load Field Dialog](#) or the [Load Information Dialog](#).

Edit Mode

Simulator offers two options for adding load fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Load Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the load to which you want to add the field. This invokes the [Load Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest load to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Load* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing load field, position the cursor anywhere on the object and right-click. This brings up the [Load Field Dialog](#). Choose , or  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

Transmission Line Display Objects

Transmission lines are represented on the onelines using multiple segment lines drawn between buses. Transmission lines may be equipped with circuit breakers that can be used to change the line's status. You can also add [pie charts](#) and [line fields](#) to transmission lines to indicate how heavily loaded the line is. The appearance of transmission lines, including line thickness and color, may also be customized.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the **Animated Flows Options** on the [Online Display Options Dialog](#).

Right-clicking on a transmission line displays the line's local menu, from which you can choose to inspect the [Branch Information Dialog](#).

Edit Mode

To add a new transmission line to the case, first select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the *from* bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the *to* bus) and double-click with the left mouse button. This calls up the [Branch Options dialog](#). The *from* and *to* bus numbers are set automatically provided the line starts and ends on existing buses. If there is just one line between the buses, the circuit number should be "1." For multiple lines between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transmission line. Enter the per unit (100 MVA base by default) resistance, reactance, total charging susceptance (that is B not B/2) for the line, and an MVA rating. Select OK to add the line. If you do not want to add the line to the case, select **Cancel**.

To modify the parameters for an existing line, position the cursor anywhere on the line and right-click. This brings up the [Branch Options dialog](#), which you can use to change various line parameters.

To change the physical appearance of the line (i.e. line color, line thickness, etc.), make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Line Fields on Onelines


Line field objects are used to show values associated with lines, transformers and dc transmission lines.

Run Mode

Right clicking on the line field gives you the option to open the [Line Field Dialog](#) or the [Line Information Dialog](#).

Edit Mode

To enter a new line field, select **Field > Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and select the line, transformer, or dc line to which you want to add the field. The [Line Field Dialog](#) will open. Enter the *near* and *far* bus number associated with the device (the default values for these fields correspond to the device on which you clicked) and the circuit number of the device. The field will display the flow value at the *near* end of the device. Enter the total number of digits to show on the screen and the number of digits to the right of the decimal point. Finally, select the desired field type.

To modify the parameters of an existing line field, position the cursor anywhere on the object and right-click. This again brings up the [Line Field Dialog](#). Select  from the [Formatting](#) ribbon group on the Draw ribbon tab to change many of the line field's display attributes.

Another way to add line fields to the oneline entails right-clicking the line and selecting *Add New Fields Around Line* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

Circuit Breakers on Onelines

Circuit breakers are used to open or close transmission lines and transformers. They are also used to place generators, loads and switched shunts in or out of service. By default, closed circuit breakers are shown as solid red squares, while open circuit breakers are shown as a green square outline. The color and filled or unfilled properties of circuit breakers can be modified in the [Online Display Options](#) under [Display Object Options](#). Only those circuit breakers that are set to use the default shape will abide by the shape specified with the Online Display Options. Circuit breakers for generators, loads, and switched shunts always use the rectangle shape. Circuit breakers that are automatically inserted for transmission lines and transformers will use the default shape unless the user specifies a different shape.

To change the status of a breaker, left-click on the breaker while in run mode. A circuit breaker directly controls the status of its associated display object. One breaker is shown on the line connecting a generator, load, or shunt to its associated bus. Two breakers are shown by default on transmission lines and transformers. Opening either of the line's circuit breakers opens the transmission line or transformer; you do not have to open circuit breakers at both ends of the line.


Note: Circuit breakers cannot be placed on dc transmission lines.

Edit Mode

By default, when you add a new display object to a oneline diagram, the necessary circuit breakers are automatically added at each end of the branch.

To add a new circuit breaker to the oneline, select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then click the line to which you want to add the breaker. The Circuit Breaker Options dialog will appear with the *from* and *to bus* numbers, and the circuit number automatically set to identify the line you selected. Specify the size for the switch and its initial status, as well as whether it will be anchored to the line so that it will move with the line. Click **OK** to add the circuit breaker, or click **Cancel** to abort the process. To add more circuit breakers to the line, simply repeat this procedure.

The shape of a branch circuit breaker can be user-specified on the Circuit Breaker Options dialog. If the option to *Use Default* is selected for the **Shape**, the shape is dictated by the default circuit breaker shape specified with the [Online Display Options](#) under the [Display Object Options](#). If a non-default shape is specified that requires an orientation, the **Orientation** option will be enabled.

To modify the parameters for an existing breaker, position the cursor anywhere on the device and right-click. This invokes the Circuit Breaker Options Dialog, from which you can change many of the breaker's parameters. Additional display settings for the circuit breaker can be accessed by selecting  from the [Formatting](#) ribbon group.

You can toggle the status of Transmission Line and Transformer circuit breakers while in Edit Mode by right-clicking on the associated line or transformer and selecting **Open Line** or **Close Line** from the local menu.

Run Mode

Circuit Breaker status can be toggled in Run Mode by Left-Clicking on a breaker to place its associated object either in or out of service. If the simulation is currently running, any effects of changing a circuit breaker's status are immediately shown on the oneline.

Line Flow Pie Charts on Onelines

The line flow pie charts are used to indicate the percentage MVA, MW, or Mvar loading of a transmission line or a transformer. The degree to which the pie chart is filled shows how close the device is to its limit (provided the device has a nonzero limit). A line flow pie chart becomes completely filled when the device's flow meets or exceeds 100% of its rating.

For pie charts for objects that are not associated with line flows, see [Pie Charts / Gauges](#).

Use the *Pie Chart Options* tab of the [Online Display Options Dialog](#) to customize various attributes of all pie charts. The tab allows you to define a warning level at which the size and color of the pie charts will change to a size and color you specify. The tab also allows you define a limit percent as well as the size and color to which to change the pie charts when their corresponding devices violate their limits. You can also specify whether the pie charts should reveal total power flow (MVA), real power flow (MW), or reactive power flow (MVR). The online display options dialog can be invoked either by selecting **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab or by right-clicking on the background of the online diagram and selecting *Online Display Options* from the resulting local menu.

Right clicking on the pie chart displays the Line Flow Pie Chart Options Dialog. This dialog allows you to view the *from* and *to bus* numbers and the circuit number of the line/transformer associated with the pie chart. You can change the pie chart's size and the MVA rating of the line/transformer associated with the pie chart.

Edit Mode

To enter a new line flow pie chart, select **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab [Ref1411868421](#) and click on the line or transformer to which you want to add the pie chart. This opens the Line Flow Pie Chart Options Dialog box. Enter the near and far bus number associated with the device (these fields default to the terminal bus numbers of the device on which you clicked), the circuit number of the device, and the desired size of the pie chart. The field will display the flow value at the *near end* of the device. Enter the size of the device. Select **OK** to insert the line flow pie chart. Otherwise, select **Cancel**.

To modify the parameters of an existing line flow pie chart, position the cursor anywhere on the object and right-click. This again brings up the Line Flow Pie Chart Options Dialog box.

Line Flow Gauges

Line flow gauges provide a way to visualize the flow of a transmission line or transformer relative to its thermal rating. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Line flow gauges in Simulator work the same way. A line flow gauge has two markings on its side, one for the designated minimum flow, and one for the designated maximum flow (the branch's rating). Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the line flow gauge, you associate it with a transmission element, and you specify its fill color and its minimum and maximum flow levels. Once the gauge has been placed on a display, it will reveal changes in its associated branch's flow by varying the height of its filled region. This tool was introduced to provide an alternative to [line flow pie charts](#).

To add a line flow gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline diagram where you would like the new line flow gauge to appear. The [Line Flow Gauge Options Dialog](#) will appear. Use this dialog to define the minimum and maximum flows for the line flow gauge, as well as its fill color and whether it should be anchored to its associated transmission element. After you click the "OK" button, the Line Flow Gauge Options Dialog will close, and the new line flow gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the flow of its transmission element changes. To modify any of the characteristics of the gauge, such as its key flow levels, fill color, and anchor setting, simply right-click on the line flow gauge to open the Line Flow Gauge Options Dialog again.

Line Flow Gauge Options Dialog

The Line Flow Gauge Options Dialog is used to define and configure a [Line Flow gauge](#). A line flow gauge is associated with a particular line and reveals the line's flow relative to specified minimum and maximum thermal flow levels. The height of the colored column in the line flow gauge indicates the line's flow relative to these markings.

The dialog has the following controls:

Number, Name, and Find

Use the **Number** and **Name** dropdown boxes, in addition to the **Circuit** field, to identify the terminal buses to which the branch desired corresponds. It may be more convenient to press the **Find** button to open the [Find Dialog](#), which allows you to specify the line by either bus names or numbers using wildcards. When you first open the Line Flow Gauge Dialog, the bus names and numbers will correspond to the transmission line object that was closest to the point where you clicked.

Minimum and Maximum

Use these two spin edit boxes in the **MW Rating values** group box to specify the minimum and maximum flow levels. These settings determine where on the gauge its two markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current transmission line, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

Anchored

A line flow gauge is said to be [anchored](#) if, when you move its associated line object, the gauge moves with it. Check the Anchored check box to ensure that the gauge will move with its associated line. Otherwise, when you move its associated line object, the gauge will stay in its current position.

OK, Help, and Cancel

Click OK to finalize your settings. This will create a new line flow gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes.

Line Flow Arrows on Onelines

There is a separate object, called a Line Flow Arrow, that can be added to a transmission line or transformer. This object is different than the animated flow arrows that can be added to all transmission branches through settings with the [Online Display Options](#). The Line Flow Arrow is a single static arrow that indicates the direction of MW flow on a transmission branch. The direction that the arrow points indicates the direction of MW flow, and there are additional options that allow the MW, Mvar, and MVA flow values to be displayed with the object.

To add a Line Flow Arrow to a transmission branch, select **Indication > Line Flow Arrow** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab while in Edit mode, then left-click on the branch to which the Line Flow Arrow should be added. The Line/Transformer Flow Object dialog will open.

The dialog contains the following options:

FromBus, ToBus, Circuit

The from bus, to bus, and circuit of the branch the arrow represents. The from bus indicates the side of the branch at which the flow will be reported. The **Find** button can be used to open a dialog that allows the selection of a branch.

Size

Size of the object.

Angle

Angle indicating how the object will be drawn. Enter this in degrees.

Anchored

Check this box to anchor the flow arrow to the branch that it represents. If anchored, the flow arrow will move accordingly any time that the branch is moved.

Auto-determine angle when anchored

Check this box to have the angle of the flow arrow determined by the branch to which it is anchored rather than using the user-specified value. The angle is determined by how the branch is drawn and the direction of MW flow on the branch. It is a good idea to use this option by default. This will ensure that the arrow is drawn correctly with the direction of flow.

Show MW, Show Mvar, Show MVA, Show Units

Check the appropriate box to show the selected flow value in text along with the flow arrow object. Check the Show Units box to include units along with the value.

Flow Text Position

Select to place any flow text either **Above Arrow** or **Below Arrow**.

Total Digits in Field, Digits to Right of Decimal

Use these fields to indicate the total digits and digits to the right of the decimal for any values that are shown in the flow text.

By default, line flow arrows will be drawn with a lime color. The color and other attributes can be changed while in edit mode. To change how the line flow arrow is drawn, select the object to change and then select **Format** from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This will open the Format Multiple Objects dialog and any attributes that can be modified will be enabled on this dialog.

DC Transmission Line Display Objects

DC transmission lines are represented on the onelines using multiple segment lines drawn between two buses. [Line fields](#) are often placed close to dc transmission lines on the oneline to indicate the power flow through the device.

Note that, unlike ac transmission lines, dc transmission lines cannot be equipped with circuit breakers.

Run Mode


Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the dc transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a dc transmission line displays the line's local menu, from which you can choose to inspect the [DC Transmission Line Dialog](#).

Edit Mode

To add a new dc transmission line to the case, first select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the bus you desire to be the rectifier bus for the line and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete the new line, place the cursor on the second bus for the line, which will serve as the inverter bus, and double-click the left mouse button. This calls up the [DC Transmission Line Dialog](#). If you successfully selected the rectifier and inverter buses, their numbers will be automatically filled in for you when the dialog opens.

The DC Transmission Line Record Dialog has four separate pages: Line Parameters, Rectifier Parameters, Inverter Parameters, and Actual Flows. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages are used to set the modeling parameters associated with the dc lines.

To modify the parameters for an existing dc transmission line, position the cursor anywhere on the line and right-click. This will provide access to the corresponding [DC Transmission Line Dialog](#), from which you can modify any of the dc line's parameters. Select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the color and/or line thickness of the dc line.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by clicking and holding the left mouse button down, dragging the vertex to a new location, and releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Multi-section Line Display Objects

Multi-section lines are represented on the onelines using a line segment with intermediate or "dummy" bus representations. These objects are not the same as a normal transmission line. They do display animated flows in run mode, but do not have pie charts, circuit breakers, or text fields currently associated with the objects.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a multi-section line displays the [Multi-section Line Information Dialog](#).

Edit Mode

Unlike transmission lines and transformers, multi-section lines CANNOT be inserted in a case by adding the multi-section line graphically. Therefore, when inserting a multi-section line object, the data for the object must already exist in the case. A list of currently defined multi-section lines is found in the model explorer under **Aggregations > Multi-Section Lines**. Right-clicking and selecting **Insert...** in the table that opens will allow you to add new multi-section line information via the [Multi-Section Line Information](#) dialog.

To insert the multi-section line object, first select **Network > MS Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the "from" bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the "to" bus) and double-click with the left mouse button. Completing the multi-section line opens the [Multi-section Line Information](#) dialog. At the bottom of this dialog are display options for the line, such as line pixel thickness, symbol size (for the series capacitor representations, if any,) anchored, and options for drawing the buses.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

Transformer Display Objects

Transformers are represented as transmission lines with two opposing coils drawn on one of the segments. The transformer's line thickness, color, and symbol segment can be customized using the [Branch Options](#) Edit mode dialog. Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the transformer. Clicking on the circuit breakers changes the status of the transformer.

[Line fields](#) are often placed close to transformers on the oneline to indicate the power flow through the device. [Transformer fields](#) are often placed close to transformers on the oneline to indicate and control their tap positions. See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the transmission line, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the **Animated Flows Options** on the [Online Display Options Dialog](#).

Right-clicking on a transformer displays the line's local menu, from which you can choose to inspect the [Branch Information Dialog](#).

Edit Mode

New transformers are inserted in much the same way as transmission lines. To add a new transformer to the case, first select **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transformer (the *from bus*) and click the left mouse button. Add more segments to the transformer by moving the cursor and clicking with the left mouse button. To complete the new transformer, place the cursor on the transformer's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Branch Options dialog](#). The *from* and *to* bus numbers for the transformer should have been set automatically. If there is just one transformer between the buses, the circuit number should be "1." For multiple transformers between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transformer, the number of the line segment in which you would like the transformer symbol drawn, and the size of the transformer symbol.

Enter the per unit (100 MVA base by default) resistance, reactance and charging susceptance for the transformer, and an MVA rating. Enter the off-nominal tap ratio and the phase shift angle in degrees. (For a transformer without tap or phase control, the off-nominal tap should be 1.0 and the phase shift angle should be 0 degrees.)

Select the appropriate Automatic Control Option. If the transformer does not have tap control, select *No Automatic Control* (this is the default). Select *AVR* (Automatic Voltage Regulation) if the transformer changes its tap ratio to control the voltage at user specified regulation bus. Select *Reactive Power Control* if the transformer changes its tap ratio to control the reactive power through the transformer. Finally, select *Phase Shift Control* if the transformer changes its phase shift to control the MW flow through the transformer. If you need any of the last three options, select the Automatic Control Options button to set the parameters associated with the automatic control.

For AVR control, enter the number of the bus whose voltage is to be controlled, the allowable range for the controlled voltage (in per unit), the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For reactive power control, the control variable is always the reactive power measured at the *from bus* (i.e., the tapped side) of the transformer. Positive flow is assumed to be going through the transformer to the *to bus*. Enter the minimum and maximum allowable flows, the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For phase shift control, the MW flow through the transformer is the controlled value. Enter the bus number of the terminal whose flow is controlled, the allowable range for the controlled flow (positive flow is assumed to be into the transformer at the terminal entered in the previous field), the minimum and maximum phase angles (typical values are -30° and 30°), and the step size in degrees (typical values are between 1° and 2°).

Select **OK** to save the values and return to the Transformer Options Dialog; otherwise select **Cancel**.

If you would like the transformer to be initially modeled as being on automatic control at the start of the case, select the **Automatic Control Active** checkbox.

If you do not want to add the transformer to the case, select Cancel.

To modify the parameters for an existing transformer, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the Line Information Dialog. This will open the [Branch Options dialog](#). Use this dialog to adjust many of the transformer's electrical properties. The [Formatting](#) ribbon group provides tools to change the transformer's color and/or line thickness.

See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

Transformer Fields on Onelines

Transformer fields are used to show field values specific to transformers, such as tap position, phase angle, and more.

Run Mode

If there is a spin control integrated with the field, you can click on the spinner to change the field's value by the associated *Delta Per Mouse Click*.

Right clicking on the transformer field gives the option to display the [Transformer Field Dialog](#) or the [Line/Transformer Information Dialog](#).

Edit Mode

To enter a new transformer field, select **Field > Transformer Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the transformer object to which you want to add the new field. This calls up the [Transformer Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the transformer on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the off-nominal tap ratio, the off-nominal tap position, or the phase shift angle in degrees.

To modify the parameters of an existing transformer field, position the cursor anywhere on the object and right-click to bring up the [Transformer Field Dialog](#). Use the tools of the [Formatting](#) ribbon group to change various display attributes for the transformer field, including its font and background color.

Three Winding Transformer Display Objects

Three winding transformer display objects do not currently have as much functionality as other display objects. They do provide a graphical representation for identifying where three winding transformers are located in the network, but the display objects do not provide access to additional control information for the transformer. Three winding transformer display objects cannot be anchored to buses like other objects can, meaning that they will not move as other objects in the display are moved around.

A three winding transformer can be added to a oneline display while in Edit mode by selecting **Network > Three-Winding Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will change the mouse cursor to a cross-hair. Place the cross-hair on the oneline background at the location where the new transformer should be placed and then left-click. This will open the Three-Winding Transformer dialog.

The Three-Winding Transformer dialog contains information about the display of the three winding transformer only. Use the [Three Winding Transformer Display](#) to view and change control parameters and other information about the transformer. The dialog contains the following parameters:

Primary Number, Name, and Area

Number, name, and area number of the primary winding bus.

Secondary Number, Name, and Area

Number, name, and area number of the secondary winding bus.

Tertiary Number, Name, and Area

Number, name, and area number of the tertiary winding bus.

Circuit

Circuit ID of the three winding transformer.

When creating a new three winding transformer on the display, the primary, secondary, and tertiary bus numbers and circuit ID must be provided that correspond to a three winding transformer that already exists in the power flow case data model. The creation of a new three winding transformer in the data model is not allowed by graphically inserting a transformer. The bus names and area numbers are provided for informational purposes only and do not have to be entered when creating a new display object.

Click the **Find** button next to the winding specifications to open a dialog that can be used to search for an existing three winding transformer in the data model.

An existing three winding transformer display object can be assigned to a new three winding transformer data object by changing the Primary, Secondary, Tertiary, and Circuit to match an existing three winding transformer data object.

Labels

Optionally, create a label to identify the three winding transformer that the display object represents. Assigning a label here will associate a label with three winding transformer in the data model. See the [Labels](#) topic for more information about labels.

This entry is an exception for this dialog. All other fields are associated with the display object only. This is the only field that is associated with the underlying data object that the display object represents.

Display Size

This is the vertical size of the object when displayed using Orientation Up or Down. This is the horizontal size of the object when displayed using Orientation Left or Right. This is in Simulator units.

Display Width

This is the vertical size of the object when displayed using Orientation Left or Right. This is the horizontal size of the object when displayed using Orientation Up or Down. This is in Simulator units.

Pixel Thickness

Thickness in pixels of the lines used to draw the object.

Anchored

This option is permanently disabled. Currently, three winding transformer objects cannot be anchored to other objects.

Orientation

Options are Right, Left, Up, and Down. The option selected determines the side of the object that the secondary and tertiary windings are on relative to the primary winding.

Tertiary Orientation

This option determines the placing of the tertiary winding relative to the secondary winding. If the **Orientation** is either Right or Left, the Tertiary Orientation options are either Up or Down. If the **Orientation** is either Up or Down, the Tertiary Orientation options are either Right or Left.

Colors

Different colors can be specified for the Primary, Secondary, and Tertiary windings. Either left-click on the color or click the **Change** button next to the appropriate winding to open a dialog that will allow color selection.

OK, Cancel, Help

Click **OK** to apply any changes and close the dialog. Click **Cancel** to close the dialog without applying any changes. Click **Help** to open this help topic.

Series Capacitor Display Objects

Series capacitors are represented as transmission lines with two opposing parallel bars drawn on one of the segments. The series capacitor's line thickness, color and symbol segment can be customized using the [Branch Options](#) Edit mode dialog. If the series capacitor branch is operating, but the series capacitor status is set to **Bypassed**, a low impedance segment will be drawn around the series capacitor symbol to indicate the capacitor has been bypassed. The capacitor status of **Bypassed** or **In Service** can be toggled in run mode if the Series Capacitor Status field is displayed on the oneline diagram. When a left-click is registered on the Series Capacitor Status field when in Run Mode, the capacitor status is toggled. Note that this is not the same as the overall branch status of **Open** or **Closed**.

Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the series capacitor. Clicking the circuit breakers changes the branch status of the series capacitor.

Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the series capacitor, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the **Animated Flows Options** on the [Online Display Options Dialog](#).

Right-clicking on a series capacitor displays the line's local menu, from which you can choose to inspect the [Branch Information Dialog](#).

Edit Mode

New series capacitors are inserted in much the same way as transmission lines. To add a new series capacitor to the case, select **Network > Series Capacitor** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the series capacitor (the *from bus*) and click the left mouse button. Add more segments to the series capacitor by moving the cursor and clicking with the left mouse button. To complete the new series capacitor, place the cursor on the series capacitor's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Branch Options dialog](#). The *from* and *to* bus numbers for the series capacitor should have been set automatically. If there is just one branch between the buses, the circuit number should be "1." For multiple branches between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the series capacitor, the number of the line segment in which you would like the series capacitor symbol drawn, and the size of the series capacitor symbol.

Enter the per unit (100 MVA base by default) resistance, reactance and charging susceptance for the series capacitor, and an MVA rating. On the Series Capacitor tab, check the box labeled **Is Series Capacitor** to indicate that the branch model is a series capacitor device.

If you do not want to add the series capacitor to the case, select Cancel. Otherwise click **OK** to add the series capacitor to the case.

To modify the parameters for an existing series capacitor, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the Line Information Dialog. This will open the [Branch Options dialog](#). Use this dialog to adjust many of the series capacitor's electrical properties.

See [Series Capacitor Information](#) for more details on modeling series capacitors.

Series Capacitor Fields on Onelines

Series capacitor fields are used to show field values specific to series capacitors.

Run Mode

If the Series Capacitor field for capacitor status is displayed, the capacitor status can be toggled in run mode when you left-click on the field.

Right-clicking on the series capacitor field gives the option to display the [Series Capacitor Field Dialog](#) or the [Branch Information Dialog](#).

Edit Mode

To enter a new series capacitor field, select **Field > Series Capacitor Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on or near the series capacitor object for which you want to add the new field. This calls up the [Series Capacitor Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the series capacitor on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the capacitor status or the series capacitance.

To modify the parameters of an existing series capacitor field, position the cursor anywhere on the object and right-click to bring up the [Series Capacitor Field Dialog](#).

Switched Shunt Display Objects

Switched shunts are either capacitors that supply reactive power to the system or reactors that absorb reactive power. Simulator represents switched shunts as a number of blocks of admittance that can be switched in a number of discrete steps or over a continuous range. Switched shunt display objects come equipped with a circuit breaker that indicates the shunt's status. If the switched shunt is closed, the circuit breaker appears as a filled red square. If the switched shunt is open, the circuit breaker appears as a green square outline. To change the status of the switched shunt, click the corresponding circuit breaker.

[Switched shunt fields](#) are often placed next to switched shunts to indicate the amount of reactive power supplied by the device. For switched shunts with such a field, you can manually increase the reactive power supplied by the device (provided its control mode is discrete) by clicking on the up-arrow associated with the device's reactive power field. Likewise, you can decrease the reactive power supplied by the device by clicking on the down-arrow. To make the up/down arrows visible, set the *Delta per Mouse Click* on the [switched shunt field](#) to a nonzero value.

Right-clicking on the switched shunt displays the [Switched Shunt Dialog](#). Use the Switched Shunt Dialog to inspect or modify the model of the switched shunt.

You can add a new switched shunt to the case in Edit Mode. Select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then click the bus where you would like to attach the device. The [Switched Shunt Dialog](#) will appear. The bus number is automatically determined from the bus to which you attached the capacitor. Enter the size, the thickness of the pen [in pixels] used to draw the device, and its orientation. The Nominal Mvar field gives the amount of reactive power the device would supply if its terminal voltage were 1.0 per unit. The Control Mode field determines whether the switched shunt has a fixed value or will vary discretely or continuously within its operating limits to maintain its terminal voltage within the voltage range specified in the Voltage Regulation field.

The amount of shunt admittance is specified in the Switched Shunt Blocks table. The columns in this field correspond to different blocks of admittance. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step. The switched shunts are always switched in the order specified in this field.

Select OK to add the device. If you do not want to add the switched shunt to the case, select Cancel.

To modify the parameters for an existing switched shunt, position the cursor on the device and right-click. This again brings up the [Switched Shunt Dialog](#). You can then change any parameter of the switched shunt. Make use of the tools in the [Formatting](#) ribbon group to change the color and/or line thickness.

To delete an existing switched shunt, use the **Cut** or **Delete** commands in the [Clipboard](#) ribbon group.

Switched Shunt Fields on Onelines

Switched Shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

Run Mode

For switched shunt fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Switched Shunt Field Dialog](#) or the [Switched Shunt Information Dialog](#).

Edit Mode


Simulator offers two options for adding switched shunt fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Switched Shunt Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then select the switched shunt to which you want to add the field. This invokes the [Switched Shunt Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest switched shunt to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new switched shunt fields entails right-clicking the bus and selecting *Add New Fields Around Switched Shunt* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing switched shunt field, position the cursor anywhere on the object and right-click. This brings up the [Switched Shunt Field Dialog](#). Make use of the tools on the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

Interface Display Objects

Interface display objects are used on the onelines to visualize the flow of power through an [interface record](#). Interface records are used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines and/or transformers, 2) total tie-lines between two adjacent areas, and 3) total tie-lines between two adjacent zones. Only area-area and zone-zone interface records can be displayed using interface display objects.

Interfaces, like transmission lines, are represented as multi-segment lines, except that they may be drawn between [area/zone objects](#) in addition to buses. Drawing interface display objects involves the same steps as drawing transmission lines and transformers. The line thickness and color of interface objects may be customized by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

[Interface Fields](#) and [Interface Pie Charts](#) are often placed close to or on the interface to indicate the power flow through the device.

Run Mode

When animation is active, the flow of the arrows on the interface object may represent either the MW flow through the interface or the currently calculated power transfer distribution factor ([PTDF](#)) pertaining to that interface. You can customize the appearance of the animated flows using the *Animated Flows Tab* of the [Online Display Options Dialog](#) available using the local menu.

Right-click on the interface to view the [Interface Dialog](#) for the interface.

Edit Mode

The quickest method of inserting new interface objects is to use the **Auto Insert > Interfaces** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Please see [Automatically Inserting Interfaces](#) for details.

To manually add a new interface object to the case, first select **Aggregation > Interface** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the starting location for the interface (usually an area/zone object or a bus object) and left-click. Add more segments to the interface by moving the cursor and the left-clicking at the end of the segment. To finish adding an interface, place the cursor on the terminal object for the interface and double-click. This then calls up the [Interface Dialog](#). Either select an existing interface, or define a new interface (see [Interface Dialog](#) for details).

Interfaces can be [anchored](#) to either area/zone objects or bus objects.

To change the shape of the interface, first left-click on the object to select it. This causes handles to appear at each vertex. You can then move any vertex by dragging it with the left mouse button down. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the interface where you would like to add the vertex.

Automatically Inserting Interface Display Objects

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

Interface Fields on Onelines

Interface field objects are used to show values associated with [interface records](#).

Run Mode

Right clicking on the interface field displays the [Interface Field Dialog](#).

Edit Mode

To enter a new interface field, select **Field > Interface Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the background of the oneline diagram where you want the field placed. This calls up the [Interface Field Dialog](#). Enter the name of the interface, the total number of digits desired in the field, and the type of field.

To modify the parameters of an existing interface field, position the cursor anywhere on the object and right-click. This again brings up the Interface Field Dialog.

InterArea Flow Options Dialog

This dialog is outdated in PowerWorld Simulator version 5 and later. See help on the [Interface Field Information Dialog](#) for the updated dialog.

When viewing an area diagram containing inter-area objects created using PowerWorld Simulator version 4.2 or older, this dialog allows you to set text fields displaying either the actual or scheduled MW flow on the inter-area object.

Interface Pie Charts on Onelines

Interface pie charts are used to graphically show the percentage loading on an [interface record](#). The amount of shaded region of the pie chart indicates how close the interface is to its limit (provided the interface has a nonzero limit). The appearance of the interface pie charts, including their color and the ability to automatically change size based upon loading level, can be customized on the [Online Display Options Dialog](#).

Right-clicking on the interface pie chart displays the [Interface Pie Chart Dialog](#). This dialog can be used to customize the size of the pie chart, or change the interface's limit.

Edit Mode

To enter a new interface pie chart, select **Pies/Gauges > Interface Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the interface object with which you want to associate the pie chart. This calls up the [Interface Pie Chart Dialog](#), which is used to customize the appearance of the pie chart.

Loading NERC Flowgates

This command reads flowgates from an Excel file and inserts them as interface records. The format for this file should be similar to the files found at <http://www.nerc.com/~oc/dfwg.html>. To access this file you must have a NERC-supplied username and password. At the time of writing we have not seen an official description of the format -- currently Simulator just mimics the format found in this file. Flowgates are used by NERC (under Policy 9) as proxies for transmission limitations and transmission service usage on the interconnected electric power network. Simulator models flowgates using the interface records. Interface records MAY include contingency elements.

The **Load NERC Book of Flowgates from Excel** option is available from the [local menu](#) of the [Interface Records](#) display.

Saving NERC Flowgates

This command writes all the interfaces to an Excel spreadsheet using the NERC flowgate format. See the files found at <http://www.nerc.com/~oc/dfwg.html> for an example of the NERC format. To access this file you must have a NERC-supplied username and password.

The **Send NERC Book of Flowgates to Excel** option is available from the **Save As** option of the [local menu](#) of the [Interface Records](#) display.

Injection Group Display Objects

Injection group display objects are used on onelines to show a summary of the total injection of an [injection group](#) from generation and load. The summary shows only the MW injection.

An injection group display object can be added to a oneline by choosing **Aggregation > Injection Group** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. After selecting this option, place the cursor at the location on the oneline where the injection group object should be placed and left-click. This will open the Injection Group Display Options dialog with the following options available:

Name

Select the name of the injection group represented by the current display object. The drop-down will contain the names of all currently defined injection groups. New injection groups cannot be created through the injection group display object.

Style

The style dictates how the object will be drawn on the oneline. The injection group object is a simple object that is either a Rectangle or Rounded Rectangle.

Caption

This option is used to determine how the injection group will be identified on the display object. Currently, only the Name can be used as the identifier.

Width, Height

Specify the size of the object through these two options.

After an object has been created, the Injection Group Display Options dialog can be opened while in Edit mode by right-clicking on the object. If in Run mode, the [Injection Groups Dialog](#) will open when right-clicking on an injection group display object.

An injection group display object uses the same [Default Drawing Values](#) as those specified for an area display object. To change how the object is drawn after it has been created, select the object to change and then select **Format** from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This will open the Format Multiple Objects dialog, and any attributes that can be modified will be enabled on this dialog. These format changes can only be done while in Edit mode.

Links to Onelines and Auxiliary Files

Run Mode

Online Links

Online links are one of the mechanisms used in Simulator to allow you to view multiple oneline diagrams on the same screen. By default, the oneline links are shown as blue rectangles surrounding a text identifier of the linked oneline diagram. Double-clicking the left mouse button anywhere within the oneline link object will display the linked oneline diagram, even if it has not already been opened.

The ability to associate multiple oneline diagrams with a single case may prove particularly helpful when dealing with large cases. If a single oneline diagram is used to display a case having many buses, the diagram may become cluttered, and it will be difficult to analyze the case in sufficient detail. The ability to associate additional onelines with the case and to call up those additional onelines using one-links can significantly enhance your view of the system.

Online Links can be used to open oneline diagrams saved in either the PowerWorld Display file (*.pwd) or Display Auxiliary file (*.axd).

Note that you can also use the **Open Oneline** command from the [Application Button](#) to open any oneline diagram (*.pwd, *.axd) file directly. See [Opening a Oneline Diagram](#) for further details.

Auxiliary File Links

Auxiliary (*.aux) file links are one of the mechanisms used in Simulator to allow you to load auxiliary files by simply clicking on a oneline object. By default, auxiliary file links are shown as blue rectangles surrounding a text identifier of the linked auxiliary file. Double-clicking the left mouse button anywhere within the auxiliary file link object will load the associated auxiliary file.

Note that you can also use the **Load Auxiliary File** command from the [Application Button](#) to open any auxiliary file (*.aux) directly. See [Auxiliary Files](#) for further details.

Edit Mode

Online Links

To add a new oneline link to a oneline diagram, first select **Background > Oneline Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at the location where you would like to display the new link. The Oneline Link Options Dialog box will open, asking you to enter the name of the file that stores the linked oneline and a caption for the clickable oneline link object. The **Browse** button can also be used to open a dialog that will allow searching for the file. Note that you can enter simply the name of the oneline file in this location or the complete directory path and name. Simulator will look in the same directory as the case file by default. However, you can also specify additional directory locations to search for oneline diagrams by opening the [Oneline Display Options Dialog](#) (via **Oneline Display Option** from the **Oneline Options** ribbon group on the [Options](#) ribbon tab, or from the [Active](#) ribbon group on the [Onelines](#) ribbon tab), switching to **Display Options** on the left, and using the **Edit Oneline Browsing Path** option to add additional search locations for oneline diagrams.


Oneline diagrams stored in either the PowerWorld Display format (*.pwd) or Display Auxiliary format (*.axd) can be opened using oneline links. If using a Display Auxiliary format (*.axd) file, the browsing path will not be used to find the file. The file must exist in the current directory or the full directory path of the file must be specified.

Auxiliary File Links

To add a new auxiliary file link to a oneline diagram, first select **Background > Auxiliary File Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at

the location where you would like to display the new link. The Online and Auxiliary Link Options Dialog will open, asking you to enter the name of the file that stores the linked auxiliary file and a caption for the clickable auxiliary file link object. Note that you can simply enter the name of the auxiliary file in this location. Simulator will look in the same directory as the case file.

Modifying Existing Links


To modify the parameters of an existing online or auxiliary file link, position the cursor anywhere on the object and right-click. This invokes the Online Link Options Dialog, allowing you to change the identity of the linked file and the caption. Select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to modify various display attributes of the online link, including font size and background color.

Document Links on Onelines

Just as you can link to other online displays from a online diagram using [online link objects](#), you can also link to documents and data on the world-wide web using Document Link Objects. When you click on Document Link Objects in Run Mode, your system's default browser will be launched to retrieve the linked URL address. This feature is not only for web URL's, though. Any file can be linked and its associated application will automatically open. This means that presentations, documents and spreadsheets can also be linked.

In addition to linking to objects online via a URL, it's also possible to put in a local file name and have that open when the document link is clicked. The behavior is the same as if you were to click on the Windows Start button, then select Run..., then type in the file name.

To add a Document link to a online diagram, select **Background > Document Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the online diagram at the position where you would link to insert the Document link. The Document Link Options Dialog box will appear, asking you to specify the world-wide web address to which to link as well as a clickable caption to display on the online diagram. Enter the requested information and press *OK* to add the Document link object to the online.

To modify the caption or address for a Document link object, right-click anywhere on its text. Specify its new parameters in the Document Link Options Dialog and press *OK*. You may also modify various aspects of its appearance, including the font size, by choosing  from the [Formatting](#) ribbon group.

Note that Document links are active only in Run Mode. Clicking on a Document link object from the Edit Mode will have no effect other than to select the object for placement, formatting, etc.

Background Lines on Onelines

The background of an oneline diagram can display added lines, polylines, and filled polygons among other items.

Edit Mode

To add a new background line, first go to the [Draw](#) ribbon tab and select **Background > Background Line** from the [Individual Insert](#) ribbon group. Then to draw a series of straight line segments on the background, follow these steps:

- Position the mouse cursor where you want the series to begin and click and release the left mouse button.
- Move the mouse to the desired termination point of the first line segment. A straight segment will follow your mouse movements. Click and release the left mouse button to complete the line segment and prepare for drawing the next line segment, or double-click if this line segment is the last segment you wish to draw.

To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where you would like the freehand shape to begin and drag the mouse to trace the shape you desire (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape you have been drawing. At this point, you can add either another freehand section or a straight line segment. When you have finished drawing in the background, double click the mouse button.

Note that background display objects composed of straight line segments display significantly faster than lines drawn freehand. Lines drawn freehand (holding down mouse button) leave a vertex point at every point on the screen, where a line composed of straight-line segments (left-clicking only where you want a vertex point) takes considerably less effort for the PC to draw.

To change the color, line thickness, and fill color associated with the line, make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex. Note that freehand lines are nothing more than a continuous series of vertices.

Background Lines Dialog

The Line Options Dialog is available while in edit mode by right-clicking on a background line and selecting **Open Dialog...** This dialog contains options for formatting the selected background line. Many of the options listed in this dialog can also be updated on the [Format](#) dialog.

Line Thickness

This field gives the thickness of the line in pixels.

Line Color

This field shows the current line color. To change the line's color, click on the box displaying the current color or click the **Change** button next to the line color. This will display the Color Dialog. Select the desired color and select OK to change the color.

Use Background Fill

When this option is checked, the selected Fill Color will be used to fill the background of the line.

Fill Color

This field shows the current fill color. To change the fill color, click on the box that displays the current fill color or click the **Change** button. This will display the Color Dialog. Select the desired color and select OK to change the color.

Immobile

When this option is checked, the background line will be forced to stay in the same position and cannot be moved by dragging it with the mouse.

List of Vertices

This grid lists the x,y coordinates for the vertices of the line in the order in which the line is drawn. The coordinates will be displayed as longitude,latitude if the [option to show coordinates in longitude,latitude](#) is selected on the [Online Display Options](#) dialog and a valid map projection is in use. The values for the coordinates can be changed by entering new values in the appropriate position in the grid.

Shift X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Shift Value** is the offset by which all of the x values will be shifted. The **Y Shift Value** is the offset by which all of the y values will be shifted. The resulting values will be the original values plus the value entered for the shift. The **Shift All Values** button must be selected for the values entered in the shift fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

Scale X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Scale Value** is the value by which all of the x values will be scaled. The **Y Scale Value** is the values by which all of the y values will be scaled. The resulting values will be the original values multiplied by the value entered for the scale. The **Scale All Values** button must be selected for the values entered in the scale fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

OK, Save and Update Display, and Cancel

OK saves the changes and closes the dialog. **Save and Update Display** saves the changes and updates the display without closing the dialog. **Cancel** closes the dialog without saving any changes.

Background Rectangles on Onelines

The background of a oneline diagram can display rectangles among other items.

Edit Mode

To add a new rectangle, first go to the [Draw](#) ribbon tab and select **Insert > Background Rectangle** from the [Individual Insert](#) ribbon group. Position the cursor where you would like to place the upper left-hand corner of the rectangle and click with the left mouse button. A rectangle having the default size is inserted. Drag the rectangle's resizing handles to resize/reshape the rectangle.

To resize or reshape an existing rectangle, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the rectangle.

To change the color, line thickness or fill color of the rectangle, first select the rectangle by clicking on it on the diagram, and then use the tools in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, or right-click on it and select **Format Background Rectangle** from the local menu.

Background Ellipses on Onelines

The background of a oneline diagram can display ellipses among other items.

Edit Mode

To add a new ellipse, first go to the [Draw](#) ribbon tab and select **Background > Background Ellipse** from the [Individual Insert](#) ribbon group. Then position the cursor where you would like to place the upper left-hand corner of the ellipse and click with the left mouse button. An ellipse having the default size and shape is inserted. Drag the ellipse's resizing handles to resize/reshape it.

To resize or reshape an existing ellipse, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the ellipse.

Right-click on the ellipse and choose **Format Background Ellipse**, or select the ellipse and use the tools in the [Formatting](#) ribbon group found on the [Draw](#) ribbon tab to change the color, line thickness and fill color of the ellipse.

Background Pictures on Onelines

The background of an oneline diagram can display a variety of objects, including lines, filled polygons, and even pictures. The latter will be discussed in this section.

Edit Mode

Simulator can insert bitmaps, jpegs, enhanced and standard metafiles, and icons from files into the oneline diagram. These pictures may either serve as a background or appear above other objects on the oneline.

To add a picture object to the oneline diagram, go to the [Draw](#) ribbon tab and select **Background > Picture** from the [Individual Insert](#) ribbon group. Then, click the oneline diagram at the point where you would like the top left corner of the graphic to be placed. The Open Picture Dialog will open, asking you to select the graphic file that you want to insert. When you select a file from the dialog, the dialog displays a preview image so that you can be sure that you are selecting the right file. When you have identified the file to insert, click *OK*. The image will then appear on the oneline. It may be resized by dragging its resizing handles.

By default, pictures are inserted at the middle stack level. Thus, they hide most other oneline display objects. To change the stack level of the picture, click it to select it, and then select the Levels/Layers button found in the [Formatting](#) ribbon group of the [Draw](#) ribbon tab. From the *Levels/Layers Tab*, select the stack level of your choice. The *Base* stack level places the picture below all other oneline display objects, while the *Top* stack level will cause the picture to obscure all other display objects.

To resize or reshape an existing picture object, click on it to select it and then drag its resizing handles.

Converting Background Lines

Background lines can be converted to other objects by splitting, merging, or converting to power system objects. This can be done while in Edit mode. To convert a background line, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. Most options are available if only a single background line has been selected and no other objects are selected. The following options are available for converting a background line:

Split Background Line

A background line can be split at a vertex or anywhere along the line. Click the point on the line where the split should be made and then right-click and select the **Split Background Line...** option from the local menu. Two background lines will result. This option is available only if exactly one background line has been selected and no other objects have been selected.

Merge Background Lines

This option is only available if exactly two background lines have been selected. The two selected background lines will be merged at the two closest ends and a single background line will result.

Convert to ac Transmission Line

After selecting this option, the [Branch Options dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting an ac transmission line from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Convert to Bus

After selecting this option, the [Bus Information dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

Converting Background Ellipses

Background ellipses can be converted to power system objects. To convert a background ellipse, right-click on a selected background ellipse on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background ellipse:

Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

Converting Background Rectangles

Background rectangles can be converted to power system objects. To convert a background rectangle, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background rectangle:

Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

Online Fields


The Online Fields are fields that can be placed on an online diagram that display values specific to the current display of the diagram, such as x and y coordinate of the mouse cursor, animation rate, zoom percentage, and x and y coordinate of the center of the diagram at the center of the screen. These fields can be inserted on a online diagram by choosing **Field > Online Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Text on Onelines

Text display objects are used to show single lines of text on the oneline.

Edit Mode

To add descriptive text to the oneline, select **Background > Text** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and click the oneline where you would like the text to appear. The Text Options Dialog will open, asking you to enter the desired text string, and the angle at which the text is to appear on the oneline diagram.


To modify an existing text object, position the cursor anywhere on the text and right-click. The New Text Options Dialog will appear, allowing you to edit the text. Use the font option  of the [Formatting](#) ribbon group to control the font and background color of the text object.

Memo Text

Memo Text display objects are used to show several lines of text on the oneline within an enclosing box.

Edit Mode

To add a Memo Text object to the oneline, select **Background > Memo Text** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then click and drag to create a rectangle on the oneline; this rectangle will contain the lines of text you enter.

To modify the contents of a Memo Text object, position the cursor anywhere on the object and right-click. To change the formatting of a Memo Text object, select the object with a left-click, then select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab; this will bring up the [Font Properties](#) tab of the [Format Multiple Objects](#) dialog box.

Generic Model Fields

The Generic Model Fields are fields that can be placed on an oneline diagram that display any value for any object in the case on the diagram. These fields can be inserted on an oneline diagram by choosing **Field > Generic Model Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

This type of field comes in handy when you want to place a variety of fields on the diagram, without inserting specific object-type fields. All fields of data are available in one location for placement on a diagram.

Supplemental Data Fields on Onelines

Supplemental Data Field objects represent information about [Supplemental Data](#) records. The information that can be represented in these fields is the same in either Edit or Run Mode.

Supplemental Data Fields can be added to a oneline while in Edit Mode from **Draw > Field > Supplemental Data Field**. After selecting this option, move the cursor to the location on the oneline where the new field should be placed and left-click. The Supplemental Data Field Options dialog will open.

The **Supplemental Data Field Options** dialog can be used to modify the properties of individual supplemental data fields on the diagram. This dialog contains the following fields:

Classification

Select the Supplemental Classification from the drop-down list of available classifications.

Name

Select the Name of the Supplemental Data record from the drop-down list of available names. This list is populated with the appropriate names based on the selected Classification.

Total Digits in Field

Total number of digits to show in the field.

Field Prefix

A prefix that can be specified and displayed with the selected value.

Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

Field Value

The current value of the field being displayed.

Rotation Angle in Degrees

The angle at which the text will be displayed on the diagram.

Type of Field

Select a field from the drop-down list of available fields or click **Find Field** to search through the list of available fields.

Select **OK** to close the dialog and save the changes, or click **Cancel** to close the dialog without saving the changes.

Pie Charts/Gauges: Lines

Pie Chart / Gauge Style

The Pie Chart Style box determines whether the [line flow pie charts](#) show the percentage loading of the line based upon the MVA flow, the MW flow, the MVR flow, the line amp/transformer MVA flow, the maximum percentage loading under contingency, or the PTDF value. The Line Gauge Style box has the same setting options as the line flow pie charts.

Always Use Limiting Flow

Typically, the flow at opposite ends of a transmission line is slightly different due to losses across the line. If this option is checked, the pie chart will correspond to limiting MVA value of the line, independent of which end of the line that value occurs. Otherwise, the pie chart will always show the MVA value at the from bus for the line.

Display Gauge Values in Percent

Check this option to display values in the line gauges as percent of loading instead of actual flow values.

Color, Size and Percentage

This section of the options dialog allows you to customize the appearance of the line flow pie charts on the diagram. The default options for the line flow pie charts are set on the MVA tab, and are always in force for at least the MVA pie chart / gauge style. However, you can choose different settings for the other styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the other five styles can be set to use the same settings as defined for MVA by checking the given option on the page. If this option is unchecked for a particular style, then any pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

Show Value Percent

When a branch's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

Normal Size Scalar, Normal Color

The standard, pre-warning fill color and scaling factor for pie charts.

Warning / Limit Scalars and Colors

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on lines exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

Make normal color the same as the line to which the pie chart or gauge is anchored (if any)

If checked, this option will cause the pie charts to assume the same normal colors as the transmission lines to which they are anchored. This may be useful if the transmission lines are colored according to their nominal voltage level.

Only Apply Warning/Limit Colors and Resizing to Monitored Elements

If this option is checked, then only those pie charts that correspond to branches selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all pie charts will obey the options prescribed in this dialog.

Some additional options are available for pie chart appearance in situations where the line itself is out of service. Clicking on the tab labeled **Open Parameters** will display these options. You can choose to have special formatting for open devices by checking the available option, and then defining the appearance options for the pie charts on the open lines. The available options include scaling the size, width of the border, border color, pie chart background color, and drawing an "X" symbol through the pie chart.

Pie Charts/Gauges: Interfaces

Pie Chart Style

The Pie Chart Style box determines whether the interface pie charts show the percentage loading of the line based upon the MW flow, the maximum percentage loading under contingency, or the PTDF value.

Color, Size and Percentage

This section of the options dialog allows you to customize the appearance of the interface flow pie charts on the diagram. The default options for the interface flow pie charts are set on the Lines MVA tab. However, you can choose different settings for the interface styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the styles can be set to use the same settings as defined for Lines MVA by checking the given option on the page. If this option is unchecked for a particular style, then any interface pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

Show Value Percent

When an interface's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

Normal Size Scalar, Normal Color

The standard, pre-warning fill color and scaling factor for pie charts.

Warning / Limit Scalars and Colors

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on interfaces exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

Make normal color the same as the line to which the pie chart or gauge is anchored (if any)

If checked, this option will cause the pie charts to assume the same normal colors as the interface to which they are anchored. This may be useful if the interfaces are colored differently on the diagram.

Only Apply Warning/Limit Colors and Resizing to Monitored Elements

If this option is checked, then only those pie charts that correspond to interfaces selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all interface pie charts will obey the options prescribed in this dialog.

Pie Charts/Gauges: Pie Chart/Gauge Styles

This section shows the Pie Chart/Gauge styles in a case information display. The user can edit several of the fields directly. Otherwise, by right-clicking on any part of the case information display and selecting on **Show Dialog** in the popup menu, the [Pie Chart/Gauge Style dialog](#), where all the values can be modified.

Pie Charts/Gauges: General Options

Show Limit Set in Header

If this option is checked, the pie chart will display a text field showing the letter of the limit set presently being used for the element.

Show Style in Footer

When checked, the pie chart will display a text field showing the style (MVA, MW, etc) that the pie chart is set to (refer to the Style property described above.)

Max. Zoom Percentage for Full Resizing

The pie charts dynamically resize when you are zooming in and out on a oneline diagram. However, you can limit the point at which the pie charts resize when zooming in by setting a zoom level in this field. This helps prevent the pie charts from getting so large that they occupy the entire screen.

Min. Pie Chart Font Size for Warning/Limit

Specifies a minimum font size for displaying text in the pie chart object. This is useful when zooming out on the diagram, to keep the text visible by setting a minimum font size.

Background Color

The background color for the pie charts is automatically set to be the same as the online background. For a different color, select Specific color from the drop-down menu, and then click on the rectangle to the right to select the color. For a clear background, select Clear from the dropdown menu.

Pie Chart Relative Font Size

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

Pie Chart Font Color

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

Pie Chart / Gauge Dialogs

There are a number of pie chart/gauge objects that can be added to one-line diagrams. These objects are inserted via the **Pies/Gauges** menu in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. The dialogs for these objects contain a common set of options that are required regardless of the object type. These common options as well as specific settings for the different object types are detailed below.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Style Dialog](#) description.

Identification

All dialogs have a section at the top that identify the pie chart/gauge object based on the type of device that the pie chart/gauge represents. The **Find...** button is available on most dialogs to open the Find tool that can be used to advanced search for a device. Specific identifying information for the different dialogs is given below:

- **Area**The area that the pie chart/gauge represents is identified based on **Area Number** or **Area Name**. To change the area that is represented, select an **Area Number** or **Area Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all areas in the case.
- **Bus**The bus that the pie chart/gauge represents is identified based on **Bus Number** or **Bus Name**. To change the bus that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Generator**The generator that the pie chart/gauge represents is identified based on **Bus Number** and **Gen ID** or **Bus Name** and **Gen ID**. To change the generator that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **Gen ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Injection Group**The injection group that the pie chart/gauge represents is identified based on the **Injection Group Name**. To change the injection group that is represented, select a new name from the dropdown box. The dropdown box is populated with all injection groups in the case.
- **Owner**The owner that the pie chart/gauge represents is identified based on **Owner Number** or **Owner Name**. To change the owner that is represented, select an **Owner Number** or **Owner Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all owners in the case.
- **Substation**The substation that the pie chart/gauge represents is identified based on **Substation Number**, **Substation Name**, or **Substation ID**. To change the substation that is represented, select a **Substation Number**, **Substation Name**, or **Substation ID** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all substations in the case.
- **Super Area**The super area that the pie chart/gauge represents is identified based on **Super Area Name**. To change the super area that is represented, select a new name from the dropdown box or click the **Find** button. The dropdown box is populated with all super areas in the case.
- **Switched Shunt**The switched shunt that the pie chart/gauge represents is identified based on **Bus Number** and **ID** or **Bus Name** and **ID**. To change the switched shunt that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all switched shunts in the case with valid area/zone/owner filters.

- **Transformer**The transformer that the pie chart/gauge represents is identified based on **Near Bus Number, Far Bus Number, and Circuit** or **Near Bus Name, Far Bus Name, and Circuit**. To change the transformer that is represented, click on the Find button to use the **Find** tool.
- **Zone**The zone that the pie chart/gauge represents is identified based on **Zone Number** or **Zone Name**. To change the zone that is represented, select a **Zone Number** or **Zone Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all zones in the case.

Type of Field

Use this option to select the field to display in the pie chart/gauge. Fields that are deemed of more common interest are listed for easy access. Any field associated with a particular device can be selected by first choosing **Select a Field** and then finding the particular field in the dropdown listing all available fields or by clicking **Find Field** and searching for the field of interest.

Current Value

Percent

This is the percent value calculated from the current value of the selected field. How the percent is calculated is based on the selection of limits and deadband and options associated with the Style. Details of the percent calculation are given in the **Pie Chart/Gauge Style Dialog** description.

Value

This is the current value of the selected field.

In-Service

When checked, the device represented by the pie chart/gauge is in-service. This is given for informational purposes only, as the status of devices cannot be changed from this dialog.

Display Information

Type

Select to display either a Pie Chart or Gauge.

Gauge Orientation

If **Type** option of Gauge is selected, set this option to draw either a Vertical or Horizontal gauge.

Anchored

When checked, the pie chart/gauge will be anchored to the display object representing the selected device.

Size, Width, Use Fixed Gauge Width/Size Ratio

Size determines the size of a pie chart and the vertical side length of a vertical gauge or horizontal side length of a horizontal gauge. **Width** is not applicable to a pie chart but determines the horizontal side length of a vertical gauge or the vertical side length of a horizontal gauge. **Width** is not enterable if the option of **Use Fixed Gauge Width/Size Ratio** is checked. This option is checked if the **Gauge Width is Always a Fixed Proportion of Size** option is checked on the Style dialog. The fixed width/size ratio is also set on the **Style** dialog.

Ignore Dynamic Sizing

When checked, any sizing based on the calculated percent will be ignored and the pie chart/gauge will stay at the defined size. Options to set the sizing based on calculated percent can be found on the **Style** dialog.

Ignore Dynamic Open Sizing

When checked, any sizing of pie chart/gauges based on the device that is represented being out-of-service will be ignored and the object will stay at the defined **Size**. Options to set the sizing for open devices can be found on the **Style** dialog.

Always Show Value (Percent)

When checked, the options set on the **Style** dialog to display the percent on pie chart/gauges are ignored and the percent value is always shown.

Style, Show Style Dialog

The **Style** defines the set of generic options that can be applied to any pie chart/gauge. An already defined style can be selected from the dropdown. Click **Show Style Dialog** to modify an existing style or to create a new style; this opens up the [Pie Chart / Gauge Style Dialog](#).

Limits

Low Limit, High Limit

Determines the low and high values from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the limits are determined from the **Style** or based on device limits. More information on how these limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.

Use Device Low Limit

When checked, the low limit will be determined from the low limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the low limit will be based on the value entered in the **Low Limit** box.

Use Device High Limit

When checked, the high limit will be determined from the high limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the high limit will be based on the value entered in the **High Limit** box.

Override with Limits from Style

When checked, the high and low limits from the selected **Style** will be used.

Zero Percent Deadband

Use Style Zero Percent Deadband

When checked, the zero percent high limit and zero percent low limits from the selected **Style** will be used.

Zero Percent Low Limit, High Limit

Determines the low limit and high limit for the zero percent deadband from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the deadband is determined from the **Style**. The deadband represents the range of values for which the calculated percent will be zero. More information on how the deadband limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.

Pie Chart / Gauge Style Dialog

The **Pie Chart/Gauge Style Dialog** is available from any of the pie chart/gauge dialogs by selecting **Show Style Dialog** or from **Options > Online Display Options... > Pie Charts/Gauges** on the **Pie Charts/Gauge Styles** tab.

This dialog lists options that can be commonly applied to pie chart/gauge objects so that these same options do not have to be set for every individual object. The defined styles can be applied to any of the new pie chart/ gauge objects that include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. These styles are currently not applicable to Lines and Interfaces. Some options are related to options that are set for individual pie chart/gauges. More information about individual object settings can be found in the **Pie Chart/Gauge Dialogs** section.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Dialogs](#) description.

Creating and Saving Styles

Style Name

Drop-down box lists all currently defined pie chart/gauge styles. Select **Add New** to create a new style. Select **Rename** to change the name of an existing style. Select **Delete** to remove an existing style.

Save AXD

Save all currently defined pie chart/gauge styles to a Display Auxiliary file.

Load AXD

Load pie chart/gauge styles from a Display Auxiliary file.

Hide all Style Objects (except in edit mode)

When checked, the pie chart/gauge will not be visible in run mode.

Show Header (Object ID)

When checked, the header of the pie chart/gauge will contain identifying information about the device that the pie chart/gauge represents.

Show Footer (Field Type)

When checked, the field type that is being used in the pie chart/gauge will be displayed in the footer of the pie chart/gauge.

Default Style

When checked, this style is one of the default styles that are defined. Default styles cannot be renamed or deleted. This is an informational field and cannot be changed by the user.

Limits and Zero Percent Deadband Values

Define the High and Low Limits to use when the option **Override with Limits from Style** is selected for the pie chart/gauge that is using this style and the High and Low Zero Percent Deadbands to use when the option **Use Style Zero Percent Deadband** is selected for the pie chart/gauge that is using this style. The deadband represents the range of values for which the calculated percent will be zero. More information is given in the [Pie Chart Parameters Tab](#) and [Gauge Parameters Tab](#) sections on how the limits are used in calculating the percent.

The remainder of the dialog is broken down into four tabs:

- [Standard Parameters](#)
- [Open Parameters](#)
- [Pie Chart Parameters](#)
- [Gauge Parameters](#)

Pie Chart / Gauge Style Dialog - Standard Parameters Tab

Normal Scalar, Normal Fill Color

These are the standard, pre-warning scaling factor and fill color for pie chart/gauges. To change the color, click on the color to open the Color dialog.

Font Color

Font color of the text displayed in the pie chart/gauge. To change the color, click on the color to open the Color dialog.

Border Width

Width of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border width also impacts the width of the lines marking the limits and deadband on the gauge.

Border Color

Color of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border color also is the color used for the lines marking the limits and deadband on the gauge. To change the color, click on the color to open the Color dialog.

Border Color Same as Fill Color

When checked, the border color will be the same color as the fill color and the color selected in the Border Color will be ignored.

Background Color

The drop-down box allows selection of three options for the background color: Online Background, Specific Color, and Clear. Online Background will color the pie chart/gauge background the same color as the selected background color for the online. Specific Color will allow the selection of a user specified color. To specify a color, click the box to the right of the drop-down to open the Color dialog. The Clear option will not use a background color and objects underneath the pie/chart gauge will be visible through the unfilled portions of the pie chart/gauge.

Relative Font Size

Modify the track bar to adjust the size of the font relative to the size of the pie chart/gauge.

Max Zoom Percent for Full Resize

Pie chart/gauges dynamically resize when zooming in and out on online diagrams. A limit can be placed on the point at which the pie chart/gauges resize when zooming in by setting a zoom level in this field. This helps prevent the pie chart/gauges from getting so large that they occupy the entire screen.

Warning/Limit Scalars and Colors

The table in this section allows the definition of different settings for pie chart/gauge size and fill color based on the percent value represented by the pie chart/gauge. Points can be added or removed from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color dialog.

Use Discrete Map

When checked, discrete colors will be used for the pie chart/gauges based on the colors specified in the table. Otherwise, colors will be blended if the percentage represented in the pie chart/gauge falls between percentages defined in the table.

Pie Chart / Gauge Style Dialog - Open Parameters Tab

Open parameters are applied to any pie chart/gauges when the device that they represent is out-of-service.

Special Formatting for Open Devices

When checked, the **Open Size Scalar**, **Open Border Width**, and **Open Border Color** options are applied when a device is out-of-service.

Use Special Open Background Color

When checked, the **Open Background Color** option is applied.

Open Symbol

When an **Open Symbol** other than None is selected, the symbol will be drawn in the selected **Open Symbol Color**.

Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab

Options given on this tab only apply to how pie charts are displayed.

Show Value Percent

When the percent of the pie chart/gauge exceeds the value specified in this field, the percent is shown as text within the pie chart/gauge. The default is 80%.

Display of Values Below the Zero Percent Deadband

The selection of this option determines how to calculate the percent represented in pie charts. Four parameters are required for determining this: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual pie charts and/or options set with the style. In general, the zero percent low limit and zero percent high limit define the deadband in which the percent is zero. The percent is then determined based on where the selected field value falls within the high limit and zero percent high limit and the low limit and zero percent low limit. The specific calculations used for determining the percent is given for the options below. In all cases if the limits are not defined correctly (i.e., none of the If..then statements below apply), the percent will be returned as zero.

- **Allow negative percent values**
 - o If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then
Percent = ((field value) – (zero percent high limit))/((high limit) – (zero percent high limit))
 - o If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then
Percent = ((field value) – (zero percent low limit))/((zero percent low limit) – (low limit))
- **Use absolute value** (i.e., percent always >= 0)
 - o Same calculations as in **Allow negative percent values** except that the absolute value of the calculated percentage is reported.
- **Low limit is zero percent**
 - o The zero percent deadband limits are ignored in this option and the percent is calculated based on the low and high limits only
 - o If (high limit) > (low limit) then
Percent = ((field value) – (low limit))/((high limit) – (low limit))
- **Treat values below zero deadband as zero**
 - o If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then
Percent = ((field value) – (zero percent high limit))/((high limit) – (zero percent high limit))
 - o If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then
Percent = 0

Pie Chart / Gauge Style Dialog - Gauge Parameters Tab

Options given on this tab only apply to how gauges are displayed.

Show Limit Labels

When checked, text labels will be drawn with the gauge showing the four limits in use.

Include Units in Limit Labels

When checked, units will be included when the limit labels are shown.

Numeric Value Type

This option allows the numeric value to either be displayed as the **Actual Value** or the calculated **Percent** value.

Display of Numeric Value

This option determines when the value of the field is displayed with the gauge. **Never** will not show the value, **Always** will always show the value, and **If Violating Warning/Limit Value** will only show the value if it is outside of the limit range that is currently in use.

Digits to Right of Decimal

This parameter determines how many digits to the right of the decimal are displayed in the limit labels and the numeric value when they are shown with the gauge.

Gauge Width is Always a Fixed Proportion of Size

When checked, the **Gauge Fixed Width/Size Ratio** is used to determine the width of the gauge as a proportion of the size (height). Note: the size of gauges is specified for individual gauges on the pie chart/gauge dialogs.

Gauge Fixed Width/Size Ratio

Specifies the ratio of the gauge width to the size when the **Gauge Width is Always a Fixed Proportion of Size** option is checked.

Percent Overhang on Top of Gauge

The overhang at the top of the gauge is the region above the high limit line. The size of this region is defined as a percentage of the total size of the gauge.

Percent Overhang on Bottom of Gauge

The overhang at the bottom of the gauge is the region below the low limit line. The size of this region is defined as a percentage of the total size of the gauge.

Gauge Fill and Percent Calculation

These options determine how the calculated percent will be displayed and how the gauge will be filled. Four parameters are required for determining these: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual gauges and/or options set with the style. In all cases if the limits are not defined correctly, the percent will be returned as zero.

Use Absolute Value for Percent

When checked, the displayed percent will be the absolute value of the calculated percent. This option does not impact how the gauge will be filled.

Gauge Fill Options (Zero Percent Reference)

This option determines how the gauge will be filled. If **Fill up or down from deadband** is selected, the fill will begin at the low or high deadband limit and continue up or down depending

upon where the field value falls within the defined limits. If **Fill up from bottom** is selected, the fill will begin at the bottom of the gauge and fill up to the field value.

Regardless of which fill option is selected, the following equations will be used to calculate the percent based upon where the field value falls within the defined limits:

If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then

$$\text{Percent} = ((\text{field value}) - (\text{zero percent high limit})) / ((\text{high limit}) - (\text{zero percent high limit}))$$

If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then

$$\text{Percent} = ((\text{field value}) - (\text{zero percent low limit})) / ((\text{zero percent low limit}) - (\text{low limit}))$$

Ignore Deadband

When checked, deadband limits are ignored. This effectively forces the fill to start from the bottom of the gauge and fill up to the field value. The percent is calculated based on the following equation and will always be reported as the absolute value:

If ((high limit) – (low limit)) <> 0 then

$$\text{Percent} = ((\text{field value}) - (\text{low limit})) / ((\text{high limit}) - (\text{low limit}))$$

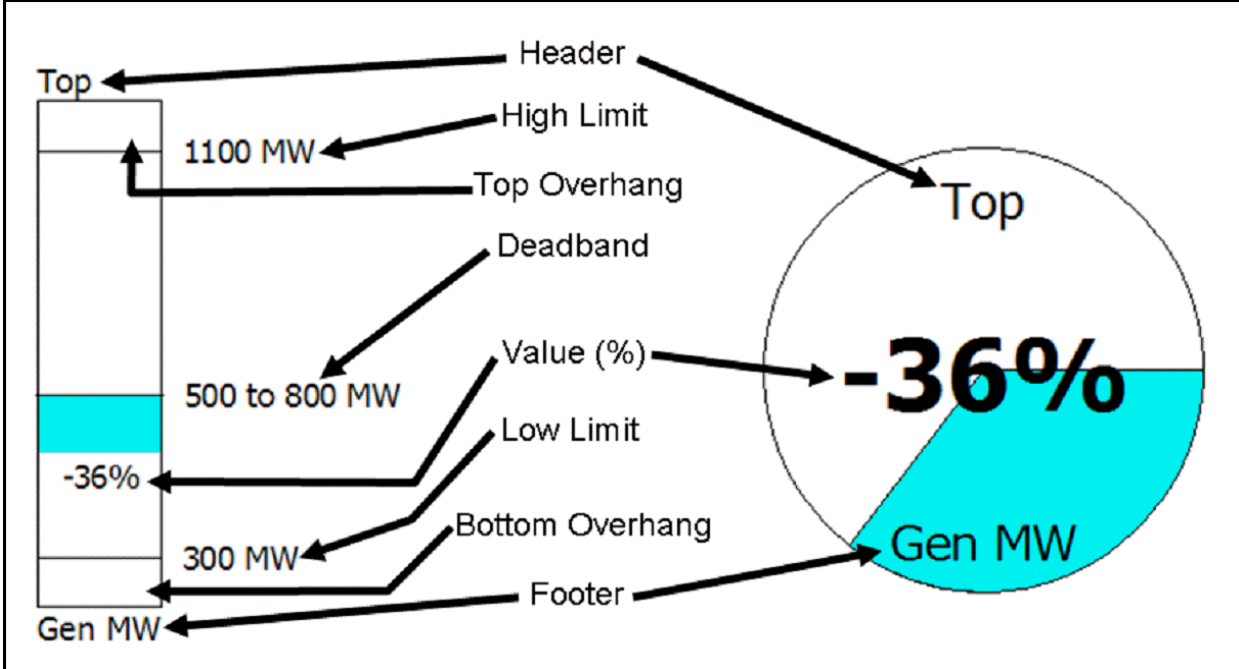
Override Low Limits with a Value of Zero

When checked, the low limit value will be set to zero when determining the fill and calculating the percent. This option is only applied if the **Ignore Deadband** check box is also checked. The percent is calculated based on the following equation and will always be reported as the absolute value:

If ((field value) > 0) and ((high limit) <> 0) then

$$\text{Percent} = (\text{field value}) / (\text{high limit})$$

Pie Chart / Gauge Example



Palette Overview

The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing. By selecting and dragging a display object name from either the **Undisplayed Neighbors** list or the **All Undisplayed** list to a location on the oneline diagram where you would like that object to appear, you can add that display object to the drawing with very little effort.

To see which elements neighboring a particular device are already on the oneline diagram and which are not, highlight a device in either the **Displayed** or **All Undisplayed** list. The **Displayed Neighbors** and **Undisplayed Neighbors** columns will list the corresponding neighboring devices for the selected device, allowing you to drag the undisplayed neighbors to the diagram if you wish.

The display object palettes, in conjunction with the of other devices, are especially useful for adding a large region of an interconnection in relatively little time.

To display one of the four available [palettes](#), select **Show Insert Palette For** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. To display the palette with the focus set to an object already displayed, right-click on that object on the diagram and select ***** Palette** (where *** is the name of the object type) from the popup menu.

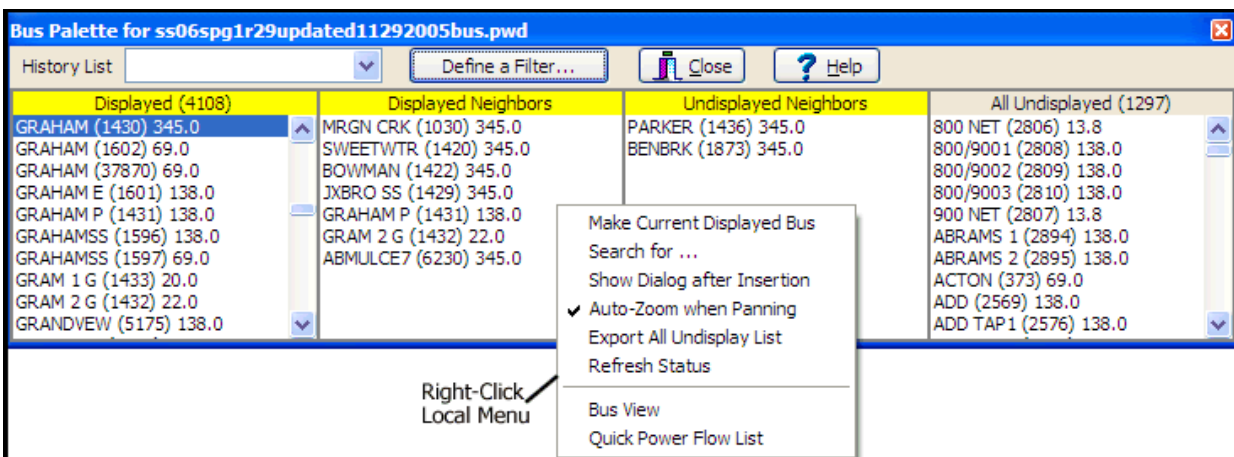
Using the Insert Palettes



The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). To open one of the palettes you must be in [Edit Mode](#) and then go to the [Draw](#) ribbon tab, and choose the **Show Insert Palette For Menu** from the [Quick Insert](#) ribbon group.

The Insert Palettes can be used in conjunction with the various auto-insert capabilities also found on the [Quick Insert](#) ribbon group. These features allow for the creation of oneline diagrams much more quickly than inserting oneline objects one at a time using the [Individual Insert](#) ribbon group.

When you choose one of the palettes the dialog which appears will look like the image below. The following image shows the Bus Palette.



Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing.

The display object palettes feature the following controls:

Displayed

Lists those objects defined in the power flow case that have already been added to the oneline diagram. When you click on an entry in the **Displayed** list, the contents of the **Displayed Neighbors** and **Undisplayed Neighbors** lists will be refreshed to identify all objects that neighbor the display object you selected. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Double click on an entry in this list to pan to it so that the selected display object appears in the center of the screen. If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View](#) to learn more about the selected bus.

Displayed Neighbors

Lists those objects defined in the power flow case that neighbor the display object selected in the **Displayed** list that have already been drawn on the oneline diagram. You can make a "Displayed Neighbor" the selected "Displayed" object by right-clicking on a display object in the **Displayed Neighbor List** and selecting **Make Current Displayed** from the local menu. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Undisplayed Neighbors

Lists those display objects defined in the power flow case that neighbor the object selected in the **Displayed** list that have not yet been drawn. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its own undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

All Undisplayed

Lists those display objects defined in the power flow case that have not yet been added to the oneline diagram, regardless of whether the object neighbors a displayed object or not. This list functions identically to the **Undisplayed Neighbors List**. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

History List

Identifies the display objects you have added to the oneline diagram using the insert palettes since the last time you opened the display object palette. Select a name from this list to display the undrawn neighbors of the corresponding display object. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Define a Filter

The **Define a Filter** button opens the [Advanced Filters](#) dialog. This dialog allows you to customize which display objects appear in the various lists. This can be helpful, for example, if you wish to add objects to the oneline that reside only in particular areas, or if you don't want objects less than a certain voltage level to be listed.

When you click **OK** on the [Advanced Filters](#) dialog, the display object palette's lists will automatically be updated to reflect the filter settings.

Close

Click **Close** when you are done using the display object palette.

Right-Click Local Menu of the Palette

There are several options available on the Right-Click

Search for : Click this option to search through the lists for a particular object. It will open the familiar [Find Object dialog](#).

Auto-Zoom When Panning : When double-clicking on the displayed objects in the palette, you will automatically be panned so that object is in the center of the screen. Check this option to also automatically zoom in/out to a reasonable level at the same time.

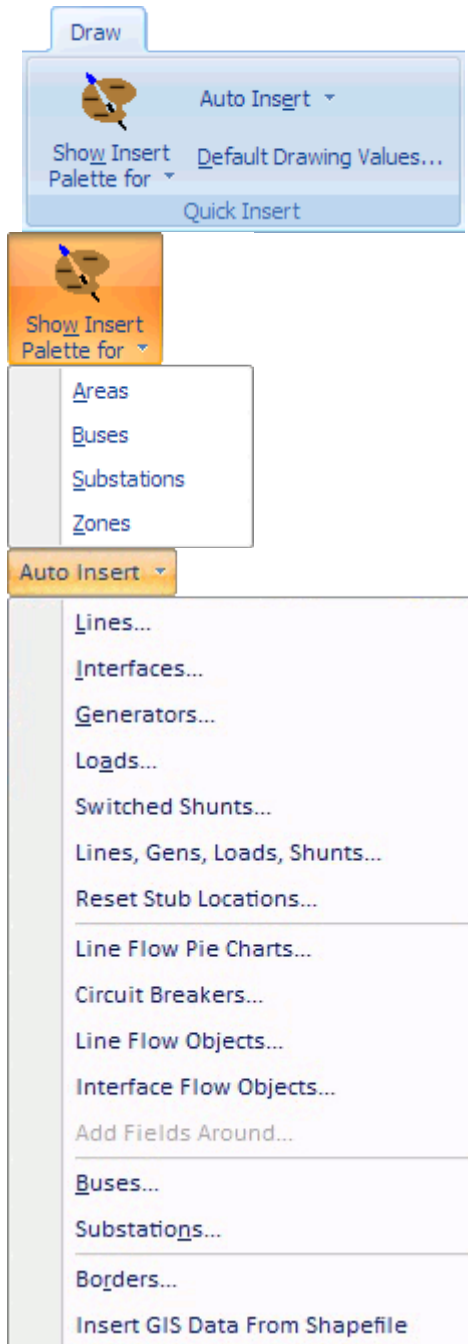
Export All Undisplayed List : Choose this option to export this list of undisplayed objects to a text file.

Refresh Status : Choose this option to ensure that the list of Displayed/Undisplayed objects is correct.

Show Dialog after Inserting : check or uncheck this option to control whether the information dialog for the object will appear when using the palette

Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic. Both of the ribbon groups are on the [Draw](#) ribbon tab. The Quick Insert ribbon group is shown as follows.



Show Insert Palette for Menu

Choosing any of the items in the Show Insert Palette for Menu will open up its respective Insert Palette. This Insert palettes show a list of all objects that are already shown on the oneline diagram as well as a list of objects which are not. For more information see the [Using the Insert Palettes](#) help topic.

Show Insert Palette for Menu

The first six options are used to automatically insert network display objects that are connected to bus objects that have already been drawn on the oneline.

Lines : Shows the [Automatically Insert Lines Dialog](#)

Interfaces : Shows the [Automatically Insert Interfaces Dialog](#)

Generators : Shows the [Automatically Insert Generators Dialog](#)

Loads : Shows the [Automatically Insert Loads Dialog](#)

Switched Shunts : Shows the [Automatically Insert Switched Shunts Dialog](#)

Lines, Gens, Loads, Shunts : Shows a dialog for auto-inserting lines, generators, loads, and shunts simultaneously.

Reset Stub Locations : Will completely redo the positioning of objects anchored to buses and substations in the manner done when auto-inserting lines, gens, load, and switched shunts.

The next four options provide methods for adding pie charts, fields, circuit breakers and line flow objects around

objects that have already been drawn on the oneline diagram.

Line Flow Pie Charts : Select this to automatically insert [line flow pie charts](#) on the transmission lines or transformers objects that are selected.

Circuit Breakers : Select this to automatically insert [circuit breakers](#) on the transmission lines or transformers objects that are selected.

Line Flow Objects : Select this option to automatically insert line flow objects on the transmission lines or transformers objects that are selected.

Interface Flow Objects : Select this option to automatically insert Interface flow objects on the interface objects. The Interface Flow Arrow Object which behaves very similar to the Line Flow Object show an arrow to represent the MW flow direction and then shown text for MW value and Mvar value with a sign relative to the MW flow direction.

Add Fields Around : Select this option to insert fields around the presently selected objects. It will then bring up the [Inserting and Placing Multiple Display Objects Dialog](#)

The follow two options provide a mechanism for auto-inserting bus or substation oneline objects. You must have geographic longitude/latitude information in order to use these options.

Buses : Shows the [Automatically Insert Buses Dialog](#)

Substations : Shows the [Automatically Insert Substations Dialog](#)

The last two options provide a mechanism for inserting background lines that represent geographic borders or from a Shapefile.

Borders : Shows the [Automatically Insert Borders Dialog](#)

Insert GIS Data From Shapefile : Shows the [Shapefile Import Dialog](#)

Automatically Inserting Buses

Simulator can automatically insert buses on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Buses** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Bus Dialog.

Note: Auto-insert buses is only for inserting bus objects on the diagram representing existing data. In other words, you cannot use the auto-insert buses routine to add new buses to the load flow model. See topics on loading data from [auxiliary files](#) or from [Excel](#) for creating new buses (or other objects) in an existing load flow case.

Insert by longitude, latitude

If you have stored latitude and longitude information with the Bus records in Simulator, you can use that information to automatically insert the buses on a oneline diagram. Select the option on the dialog for finding bus locations based on **Longitude and Latitude stored with data records**. When selected, the [Map Projection](#) option will become enabled. Choose the type of projection you would like to be used for placing the buses on the diagram.

Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows:

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Bus Number, X location, and Y location. If you are reading longitude, latitude, then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the buses on the oneline diagram at the X, Y locations given. If you specified that the file contained longitude, latitude information, then select the [Map Projection](#) to use when converting the longitude, latitude values from the file. The buses will be drawn according to default bus object information defined in the Default Drawing Values for New Objects dialog.

If you would like Simulator to **Autoinsert transmission lineswhen finished** with the auto insertion of your buses, check the box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert bus only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the buses according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the buses.

Automatically Inserting Transmission Lines

The Automatic Insertion of Lines/Transformers Dialog is used to automatically draw transmission lines and transformers on the oneline diagram between existing bus display objects. Only branches that are already defined in the power flow case can be added automatically; if you need to define a brand new branch, see [Transmission Line Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert lines and transformers automatically, you must first have drawn the buses for each end of the device. Simulator then draws the branch display objects connecting the buses for each transmission line/transformer in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Lines** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw line and transformer display objects between terminals whose nominal voltages meet or exceed the minimum kV level specified in this field. For a branch object to be drawn, either one of its terminals must satisfy this criterion. This option is useful for suppressing the automatic addition of generator step-up transformers if that kind of detail is not warranted.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of transmission line stubs, text fields, circuit breakers, and pie charts are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

If this box is checked, the default fields associated with the transmission line will appear around the transmission line.

Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as lines will also be automatically inserted with the real transmission lines.

Use Only Selected Buses

Instead of having Simulator automatically insert line and transformer display objects throughout the oneline diagram, you can force it to insert the new objects only between the [bus display objects](#) that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Insert Pie Chart for Lines with No Limit and Bus Ties

If this option is checked, Simulator will add [pie chart objects](#) to the lines that have no given limit or are bus ties as they are auto-inserted. Typically if a line is a bus tie or has no given limit, it is meaningless to include a pie chart on the element, since no relevant information about the transmission element can be gained from the pie chart object.

Insert Multi Section Lines

Check this option if you wish for Simulator to automatically insert lines designated as [multi-section lines](#) as well.

Identifying Bus Ties

This area deals with lines used as ties between breakers. These lines are modeled as zero impedance connections. The identification of a branch as a bus tie depends on the value specified as the **Maximum P.U. Impedance for Bus Ties**. Branches with total P.U. impedance below this value will be considered bus ties when auto-inserted.

In this area two choices are given for how to insert the bus tie breakers: **Do not insert stubs for bus ties** and **Only insert a single circuit breaker**. The first choice allows you to decide if line stubs will be drawn when the tie breaker is inserted. The second sets whether or not only one circuit breaker is inserted on the tie breaker. This could be useful for determining real lines from bus tie breakers.

Automatically Inserting Generators

The Automatic Insertion of Generators Dialog is used to automatically draw generators on the oneline diagram on existing bus-display objects. Only generators that are already defined in the power flow case can be added automatically; if you need to define a new generator, see [Generator Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert generators automatically, you must first have drawn the terminal bus for each device. Simulator then draws the generator display objects connected to the buses for each generator in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Generators** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw generator display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a generator object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the generator will appear around the generator, such as voltage, name, and/or MW.

Insert Equivalenced Objects

This field is not used when automatically inserting generators.

Use Only Selected Buses

Instead of having Simulator automatically insert generator display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Loads

The Automatic Insertion of Loads Dialog is used to automatically draw loads on the oneline diagram on existing bus display objects. Only loads that are already defined in the power flow case can be added automatically; if you need to define a brand new load, see [Load Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert loads automatically, you must first have drawn the terminal bus for each device. Simulator then draws the load display objects connected to the buses for each load in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Loads** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw load display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a load object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the load will appear with the loads, such as MVAR and/or MW.

Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as loads will also be automatically inserted with the real transmission lines.

Use Only Selected Buses

Instead of having Simulator automatically insert load display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Switched Shunts

The Automatic Insertion of Switched Shunts Dialog is used to automatically draw switched shunts on the oneline diagram on existing bus-display objects. Only switched shunts that are already defined in the power flow case can be added automatically; if you need to define a brand new switched shunt, see [Switched Shunt Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert switched shunts automatically, you must first have drawn the terminal bus for each device. Simulator then draws the switched shunt display objects connected to the buses for each switched shunt in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Switched Shunts** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

Minimum kV Level

Simulator will automatically draw switched shunt display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a switched shunt object to be drawn, its terminal must satisfy this criterion.

Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

Insert Text Fields

When this box is checked, the default fields associated with the switched shunt will appear around the switched shunt, such as nominal MVAR.

Use Only Selected Buses

Instead of having Simulator automatically insert switched shunt display objects throughout the oneline diagram, you can force it to insert the new objects only at the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

Automatically Inserting Interface Display Objects

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

Automatically Inserting Substations

Simulator can automatically insert substations on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Substations** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Substation Dialog.

Insert by longitude, latitude

If you have stored latitude and longitude information with the Substation records in Simulator, you can use that information to automatically insert the substations on a oneline diagram. Select the option on the dialog for finding substation locations based on **Longitude and Latitude stored with data records**. When selected, the [Map Projection](#) option will become enabled. Choose what type of projection you would like to be used for placing the substations on the diagram.

Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Substation Number, X location, and Y location. If you are reading longitude, latitude then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the substations on the oneline diagram at the X, Y locations. If you specified that the file contained longitude, latitude information, then select the [Map Projection](#) you would like to be used when converting the longitude, latitude values from the file. The substations will be drawn according to default substations object information defined in the Default Drawing Values for New Objects dialog.

Finally, if you would like Simulator to **Auto insert transmission lines when finished** with the auto insertion of your substations, check this box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert substation only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the substations according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the substations.

Automatically Inserting Borders

PowerWorld Simulator allows you to automatically insert geographic borders, which PowerWorld Corporation has drawn. These include the states in the United States of America and several international borders as well. You may also define a border in a text file and insert this User-Defined border. It is highly recommended that you create a diagram and insert borders before you begin adding power system objects to the diagram.

To bring up the **Auto Insert Borders Dialog**, go to the [Draw](#) ribbon tab and choose **Auto Insert > Borders** on the [Quick Insert](#) ribbon group.

Options

The Options tab allows you to set characteristics of the border lines when they are inserted on the oneline diagram. Most of these options are available from the Format menu in Edit Mode, and can be changed for a border by selecting the border(s) in Edit Mode and then selecting the appropriate option from the Format menu to change the desired option(s).

Make border lines immobile

This option is very useful for preventing you from inadvertently moving the border lines in Edit Mode once they have been placed.

Automatically link county and state borders to supplemental data

Background lines created from inserting borders can be linked to [Supplemental Data](#) records for purposes of further identifying these objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from inserting the borders. This option is only available when inserting United States borders.

When this option is checked, background lines created for the borders will be automatically linked to Supplemental Data records. If necessary, Supplemental Data records will be created with the Classification being either *State* or *County*, depending on the Border Type selected when inserting United States borders, and the Name being the name of either the state or county.

Line Options

Choose the line thickness and line color for the border lines.

Fill Options

Choose the fill options if you wish to "fill in" the border regions with a background fill color.

Stack Level

Choose the stack level for the border lines. The stack level affects which objects appear "over" or "under" other objects on the diagram. Typically background lines are either placed in the Background or Base levels, so that they appear beneath power system objects on the diagram. Further we would recommend that if you are using background fill colors for the borders that you choose a stack level of Base. The only difference between the Background and Base levels is that the Base level objects can be right-clicked on in Run Mode and allows the default diagram popup menu to appear. Right-clicking on Background level objects in Run Mode will not display the default popup menu.

Layers

When Borders are automatically inserted on a diagram using this tool, a *Borders* [screen layer](#) will be added to the layer drop-down box, if a *Borders* layer does not already exist for the oneline diagram.

You can choose to leave the borders in the standard *Default* layer, to place the borders in the new *Borders* layer, or you can [create a new custom layer](#) by clicking the **Define Layers** button. If no default borders layer has been previously established (borders were previously inserted on the oneline), the *Borders* layer or selected user created layer will be set as the default Borders layer. The default Borders layer is different than the standard *Default* layer for objects. The default Borders layer information is stored in the system registry, and is only used to recall which layer borders were assigned to the previous time borders were automatically inserted.

Apply Default Drawing Values

Click this button to apply the Background line settings from the [Default Drawing Values](#).

Border File Path

Designate the border file location.

PowerWorld Library

Pre-Defined Selections

Choose between North America, USA State Borders, Canadian Province Borders, or Entire World to insert the selected region. These options make things quicker than selecting from the individual menus if entire countries or continents are to be inserted.

Once you have selected the regions that you want to insert, Click **OK**.

United States

This tab allows you to select states in the United States that you want to insert. (Note: to select several states to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These states will be placed on the screen such that as you add new states they will be placed geographically appropriately.

Include Hawaii and Alaska: Select all or any of these items to place these states on the screen.

Border Types: Select the type of border. *State* will only show the state border and the *County* will show the state border as well as the counties of the state.

Once you have selected the states you want to insert, Click **OK**.

Canada

This tab allows you to select provinces in Canada that you want to insert. (Note: to select several provinces to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These provinces will be placed on the screen such that as you add new provinces they will be placed geographically appropriately.

Once you have selected the provinces you want to insert, Click **OK**.

World

This tab allows to insert borders which PowerWorld Corporation has acquired for countries around the world. PowerWorld Corporation will continue to add more border files as we receive them. Note: if you have drawn a oneline which contains a geographic border you would like us to include in future version of Simulator, please contact us at support@powerworld.com and we'll add it to our next release.

[Map Projection](#)

This setting is important for anyone including border files outside of North America. By default, Simulator draws borders using a simple conic projection referenced to North America. This can result in border files of other countries being drawn incorrectly if based on this reference. To correctly

draw borders of other countries around the world, select the Entire World setting, which uses a Mercator projection.

User-Defined Borders

This tab allows you to read in a border from a text file that you create. The first row of this text file is a comment row and is ignored when reading it in. After this it reads in the description of each background line. This description starts with the number of points in the line followed by a list of x, y coordinates. The file ends when the number of points for a line is read as -1. An example file follows:

Comment row

5

59, 60

10, 20

40.3, 95.20

89.3, 22.11

79.5, 34.56

45

40, 66

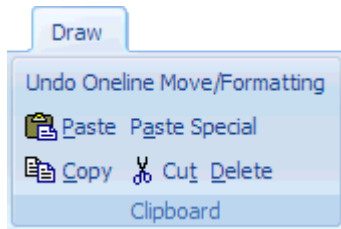
etc...

-1 this signifies the end of the file

To read the data, first specify the file name and location containing the data using the Browse button. Next indicate if the file contains data in Longitude, Latitude or in converted Simulator X,Y coordinates. If the data is Longitude and Latitude, then also specify if the map projection to use should be North America (simple conic) or Entire World (Mercator).

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the oneline. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different oneline diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Select by Criteria Dialog



To open the Select By Criteria Dialog go to the [Draw](#) ribbon tab and choose **Select By Criteria** from the [Select](#) ribbon group.

The Select By Criteria Dialog provides a way of selecting objects that meet a specific set of criteria. The criteria may include Area Numbers, Zone Numbers, Voltage Levels, Zoom Levels, and Object Type.

Use the dialog's controls to specify the selection criteria. Use the **Area** and **Zone** fields to select the areas and zones in which you want to select display objects. [Ranges of area and zone numbers](#) can be entered in the usual way, or you can check the **All Areas** or **All Zones** boxes to select all areas and all zones. You can also use the **Area** and **Zone** tab pages to check or uncheck areas or zones in which you want to select display objects.

Specify the max and min voltage levels for selected objects using the boxes that are provided, keeping in mind that all voltages are in kV.

Specify the [layers](#) for selected objects in the **Layers** tab. If not all layers are selected, the **Layers Range** option will be selected. If the **All Layers** option is clicked, then all the layers will be selected.

Next, select the type of object in which you are interested from the supplied list. To select items from this list, check the box next to the type of field(s) you desire. Display object types that are followed by a right-arrow, such as *Area Fields*, are expandable, meaning that they have several associated subtypes. If you checked a field type that has subtypes, you can highlight the type in the list to see the available subtype list. You can then more specifically select subtypes that you wish to include. By default, all subtypes of a general object type are selected. You also have the option for the list to display all fields available for a particular object type, or to display only the most commonly selected fields.

You can also choose to associate an [advanced filter](#) with a particular type of object in the list for selecting only objects of that type that meet the filter. To do so, right-click in the Filter column next to the object type for which the filter applies, or click on **Add/Modify Filter...** Some objects cannot be associated with an advanced filter, and the option will be disabled when that is the case for an object selected in the dialog.

Checking the option **Group By Object Type** will consolidate all the items by categories. All subtypes will automatically be included as a consequence. The filters are still available.

If the box **Only Show Objects in Display or Already Selected** is checked, only the items or categories (depending on whether the Group By Object Type option is checked) of objects existing or selected in the respective oneline diagram will be shown in the supplied list.

The buttons **Check All** and **Uncheck All** are useful to select/unselect all the items/groups in the list. The button **Check Only Text Fields** is available only when the Group by Object Type is unchecked, and when clicked on all the items that have text associated to them are selected, and the rest of the items are unselected. Clicking on the **Reset To Defaults** button will remove all the current selections and reset them to the default settings.

If you wish to select objects that *do not* have the chosen criteria, then check **Select All Except What Meets the Above Criteria**.

If you wish to only select objects that are currently visible on the diagram (due to layering, etc.), check the box labeled **Select only currently visible objects** to apply the criteria settings only to objects on the diagram that are currently visible.

If the Select by Criteria Dialog is open and some display objects have already been selected, an advanced selection option becomes available. If the box labeled **Use as a filter on presently selected objects** is checked, the criteria chosen in the Select by Criteria Dialog *will only* affect the previously selected objects. In other words, only the objects that were previously selected AND that

match the chosen criteria will remain selected when the OK button is pressed. For example, you could use all the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects on the oneline. Then using the Select by Criteria Dialog, choose the Bus criteria and check **Use as a filter on presently selected objects** to select only the buses from the rectangular selected group of objects.

Once you have selected the criteria and pressed *OK*, all objects that satisfy ALL the criteria will be chosen. If the Select All Except What Meets the Above Criteria is unchecked, then the selected objects will lie within the specified areas AND zones, the specified voltage level, and the specified zoom levels, and will be of one of the drawing types specified.

You can save your settings for your Select by Criteria session with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This allows you to re-open the Select by Criteria dialog, and quickly recover settings you may have previously used. Use the **Rename** to rename an already saved set, or **Delete** to remove a previously saved set of criteria.

If you have saved different criteria sets, you can export them all to an [Auxiliary file](#) by choosing **Save to AUX file**. The **Load All from AUX file** will load an entire auxiliary file, regardless of whether it contains Select by Criteria settings or not.

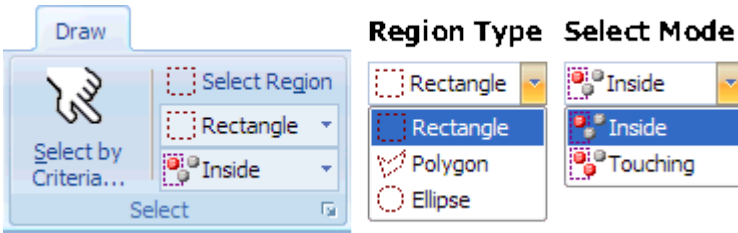
If you wish to clear all settings you have modified on the dialog, click the **Reset to Defaults** button.

Using this dialog in conjunction with the [Format Selection Dialog](#), can be a very fast and easy way to customize your displays.

NOTE: When choosing to select all tie lines, all objects associated with the tie line will also be selected. This includes all line fields for the line as well as the two buses that are the endpoints of the line.

Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On oneline diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is Polygon, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

Select Mode

This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key while clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same effect as click on the **Select Region** button and selecting a region on your oneline

Grid/Highlight Unlinked Objects

The Grid/Highlight Unlinked Objects tab is only available in Edit mode. It controls the appearance of a grid in the oneline. A grid is not visible by default, but can be setup from the Grid/Highlight Unlinked Objects page of the [Online Display Options](#) dialog. Additional options are available from this dialog for disabling anchors and highlighting unlinked objects on oneline diagrams.

Snap Objects to Grid

If checked, new objects placed on the oneline will be aligned with the grid; the grid does not need to be visible on the oneline to align objects to it.

X Grid Spacing, Y Grid Spacing

These determine the horizontal and vertical spacing of the grid.

Display Grid Lines on Oneline

If checked, the grid lines will appear on the oneline

Horizontal Show Every, Vertical Show Every

These numbers determine the density of the visible gridlines; for fewer lines enter higher numbers.

Gridline Color

Click in the rectangle to choose the color of the gridlines. The default is grey.

Anchor Options

A check box is available for temporarily disabling anchors on a oneline diagram. This option will disable the anchor properties of all objects on the oneline diagram.

Do not prompt regarding relinking objects after dragging

Check this check-box to avoid being prompted whether to relink a graphical object after this has been dragged.

Highlighting of Unlinked Objects

Highlight Unlinked Objects with Color

Checking this box allows for any objects on all open oneline diagrams that are currently not linked to any data in the case to be highlighted using the highlight color. To change the highlight color used, left-click in the color box to choose a different color.

Minimum Highlighted Object Pixel Size

The minimum size, in pixels, of the highlight image. This is to prevent the highlight from being unnoticeable when the zoom level is very low.

Extra Width for Highlighted Lines (pixels)

Sets an extra width parameter, in pixels, for highlighting line objects. This will make line object highlights appear wider than other highlighted objects.

Display Unlinked Elements in Run Mode

This option allows the highlighting of unlinked elements when you switch to run mode. By default the highlighting of unlinked elements is restricted to edit mode, to avoid confusion in run mode, particularly when drawing [contours](#).

Using the Online Alignment Grid

The online alignment grid allows for buses and other objects to be easily aligned when placed onto a oneline diagram. The alignment grid for a oneline diagram can be configured in the Oneline Display Options dialog under the [Grid/Highlight Unlinked](#) page. You can open this display by selecting **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.

When the alignment grid is enabled most objects will snap to the grid while being moved on the oneline. Depending on whether snap to grid is enabled or disabled, objects can individually be made to do the *opposite* of the snap to grid setting by holding down the Alt key while dragging the object.

Setting Background Color

The Background Color Dialog changes the background color for the display. To view this dialog, select **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab, or right-click in the background of the oneline and select **Online Display Options** from the local menu. Switch to the [Display Options](#) tab, click **Change Background Color**, or click on the colored rectangle to bring up the Color Dialog, which you can then use to select the new background color. Click OK to register the new color. The Online Display Options dialog will provide a preview of the color you selected. To define this color as the default background color to use in all new onelines you create, click the **Set as Default Background Color**. Finally, click OK to save your color selection.

Zoom, Pan, and Find

The Zoom, Pan, and Find Options Dialog is used to specify a desired zoom level and screen location or to locate a particular bus or area/zone on the oneline. The dialog can be opened by right-clicking on the background of the diagram and choosing **Pan/Zoom Control** from the oneline popup menu, or by clicking on the **Find** button in the [Zoom](#) ribbon group on the [Onelines](#) ribbon tab. The dialog has the following fields:

Find Objects on Oneline

The Find Object on Oneline section of the dialog is used to pan to any desired [bus object](#), [area/zone object](#), [branch object](#), [interface object](#), or [substation object](#) on the oneline. The dialog makes use of the advanced find functionality of Simulator to locate a device.

Only Include Objects Visible at Current Zoom Level

If checked (the default), then the **Object ID** combo box only lists those objects that are visible at the current zoom level.

Object Type

Select the type of object to find: *Buses, Areas or Zones, Interfaces, Substations and Lines/Transformers.*

Sort By

Specify whether you are entering the object by its number or by its name by choosing either **Sort by Name** or **Sort by Number**. This option also determines how the entries in the list of objects are sorted (either by number or by name).

Object ID

Enter the object's number or name (depending on the sort type chosen) or select the object from the list.

Allow Auto Updating on Selection

Check this box to automatically pan to the specified object. The object will be located at the center of the oneline diagram.

Pan to Object on Oneline

Click this button to pan to the specified object if the Auto Updating box is not checked. The object will be located at the center of the oneline diagram.

Auto-Zoom when Panning

Check this box to automatically change the zoom when panning to the specified object.

Zoom / Pan

The Oneline Zoom / Pan tab on this dialog is used to allow the user to specify either a new zoom level and/or screen center and to define these as the new display default settings. The zoom level and screen center can also be changed from the keyboard. Please see [Online Zooming and Panning](#) for details.

Zoom Level

Enter the desired percentage zoom level (nominal is 100%).

Horizontal, Vertical

Enter a desired location for the center of the screen. The nominal screen center is 50 horizontal and 50 vertical.

Pan/Zoom to New Location

Changes the screen center and zoom level to the values specified in the above fields.

Restore Default Values

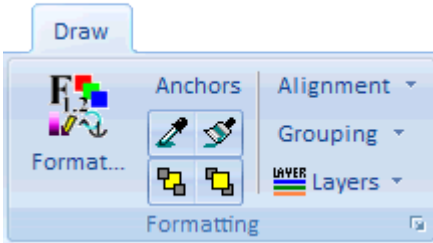
Resets the zoom level and screen center to the default values.

Default Drawing Values

The [Default Drawing Options Dialog](#) allows you to see and change the various default values used to create new objects. To show this dialog select **Default Drawing Values** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.

Formatting Ribbon Group

The **Formatting** ribbon group contains options for formatting the selected objects on your active oneline. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.



Format Buttons

Clicking this button will open the [Format Dialog](#). The dialog which opens will have five tabs as follows.

Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the oneline diagram.

Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.

Anchors

Invokes a command to [refresh anchors](#) on all oneline objects.



Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



Send to Back, Bring to Front Buttons

After selecting a group of oneline objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

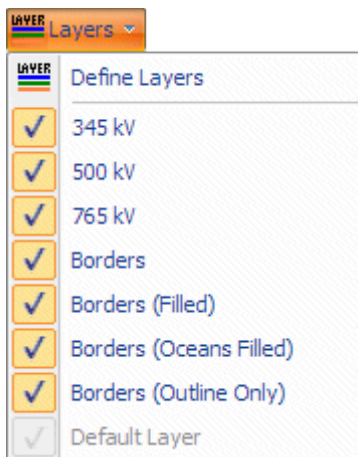
Grouping Menu

Grouping is only available for background oneline objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



Format Multiple Objects

The Format Multiple Objects dialog features five tabs of controls for modifying the display attributes of selected object(s). Even though the dialog is very useful for formatting multiple objects, there is no requirement than more than one object be selected to use the formatting dialog. To open the Format Multiple Options dialog you must first use some of the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects. Next, right-click on any of the selected objects and select **Format Selection** from the local menu. You may also bring up the Format Multiple Objects dialog by going to the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

The options on the following tabs will be updated to reflect the types of objects selected. All options will not be available for all objects. If multiple objects of different types are selected, options will only be available if they are relevant for any of the types of objects selected.

Line/Fill

The [Line/Fill tab](#) is used to change the line size, color, and style and the fill color with which the selected objects are drawn.

Levels/Layers

The [Levels/Layers tab](#) is used to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize.

Display/Size

The [Display/Size tab](#) controls the size and orientation of the selected objects.

Font

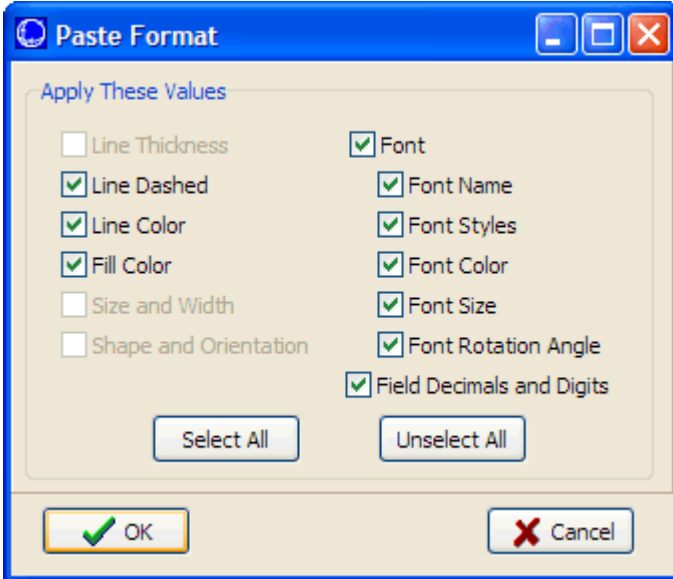
The [Font tab](#) is used to change the font used in text/font objects.

Field

The [Field tab](#) is used to change selected fields on the diagram to a different field designation. If all of the selected fields are for the same type of object and field, you can select a different field to be represented by the selected objects from the **Field** selection. You can also set the options for including a suffix, the total digits in the field for numeric values, and the number of digits to the right of the decimal.

Paste Format Dialog

The Paste Format Dialog appears when you use the Paste Format button on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. The dialog looks as follows



On this dialog check the attributes you would like to paste to the selected objects. Note that attributes which are disabled in the Paste Format Dialog represent one of two things


- The attribute was not relevant to the objects whose format was copied
- The attribute was not the same for the objects whose format was copied

Font Properties

The Font Tab allows you to define the font for the selected objects by choosing Font type, size, color and effects. The page also allows you to control the default display font using the **Make Default** button.

By clicking on **Make Default**, the current font name, size, and effects will be set as the default font for the display. This has the same effect as opening the [Default Drawing Values](#) and setting the default font.

Line/Fill Properties

The Line/Fill tab of the [Format Multiple Objects](#) dialog is used to customize the line size/color and the fill color of selected objects. You can view this tab by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab after object(s) have been selected on the oneline. This tab's controls can be used to change the appearance of most, but not all, screen objects. Only options that are relevant for the types of objects selected will be enabled. The Line/Fill tab has the following fields/commands:

Line Thickness

Thickness of the line in pixels. Line can mean either an actual transmission line or background line object, the lines used to draw objects like loads and switched shunts, or the border line that surrounds objects like generators and buses.

Dashed

Allows setting a line to appear as a dashed line on the diagram. There are three types of line dashing to choose from in the drop down menu for this option.

Line Color

This field shows the current line color. To change the line's color, click on the box displaying the current line color, or click the **Change** button next to the Line Color box. This displays the Color dialog. Select the desired color and select OK to change the color, or click Cancel if you do not wish to change the color.

Line Color 2

The Line Color 2 field applies only to transformer and generator objects. Check the **Use Color 2** field to implement use of the color and specify the color through the color box. The color can be changed in the same manner as described above for **Line Color**.

Transformers can be represented by different colors on each side of the transformer coils. This is commonly used to color match each side of the transformer with the color used to represent the voltage level of the transmission system on each particular side of the transformer. However, you can customize the colors for selected transformers by modifying them here. The Line Color 2 field ALWAYS applies to the high voltage side of the transformer.

The rotor image selected for generator objects can be filled with Line Color 2.


Use Background Fill

Click on this box to toggle whether or not to fill the background for the selected objects with the selected background fill color. Text objects such as [Text](#), [Bus Fields](#), [Generator Fields](#), [Load Fields](#), [Switched Shunt Fields](#), [Line Fields](#), [Transformer Fields](#), [Area Fields](#), background objects such as [Background Lines](#) and [Background Rectangles](#), and [generator objects](#) can be filled.

Fill Color

This field shows the current fill color. To change the fill color, click on the box that displays the current fill color, or click the **Change** button next to this box. This displays the Color dialog. Click on the desired color and the select OK to change the color, or click Cancel if you do not wish to change the color.

Levels/Layers Options

The Levels/Layers Tab is used to customize the stack level or layer an object is contained in. To set layers for objects, select the objects on the oneline diagram (edit mode only) and then choose . The Levels/Layers Options page of the [Format Multiple Objects](#) dialog will be displayed, with the following settings available.

Stack Level

An object's stack level dictates what objects it will appear above, and which objects it will appear below on a oneline diagram. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore anything with a stack level of Middle, Background or Base will appear underneath pie charts and circuit breakers on the oneline diagram. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the diagram. You can toggle which elements within the same stack level at the same location is visible using the Send to Back and Bring to Front buttons in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

Layers

Layers are different than stack levels in that layers are designed to allow the user to filter the elements visible on a diagram (in run mode). Initially all diagram objects are in the same layer, called Default. This default layer cannot be modified or deleted. However, you can introduce additional layers using either the **Add New...** or **Define...** buttons. Clicking on **Add New...** will open a dialog to name the new layer, and will automatically add the new layer to the case and change the Layer name in the drop down list to the new layer. Clicking on **Define...** will open the [Screen Layers](#) list display, which will allow you to manage the full list of layers defined for the diagram. Once layers have been defined, you can choose which layer the selected objects belong to by choosing the Layer name from the drop-down list by clicking on the down arrow to the right of the layer name.

Settings for resizing when zooming

You can modify the maximum and minimum zoom levels at which the selected text fields will no longer resize. Thus when zooming in or out on the diagram, text fields will resize according to the zoom level until the minimum or maximum zoom level are reached. At that point, text fields will no longer resize, but will stay fixed at their current size as you continue to zoom.

Maintain Fixed Screen Location (do not pan)

This option is available for text fields. When checked for selected text fields, these text fields will no longer pan when you pan the rest of the diagram. This allows you to place text that will always be visible, regardless of what part of the diagram you are observing. You can still move the individual text field itself in edit mode.

Maintain Fixed Screen Size (do not resize on zoom)

This option is available for text fields. When checked for selected text fields, these text fields will no longer resize when the zoom level of the diagram is changed. This allows you to place text that will always be visible at a constant size, regardless of what zoom level you are currently observing on the oneline diagram. You can still select the text fields themselves in edit mode and change their font size.

Screen Layers

The Screen Layers display can be invoked from either the [Levels/Layers](#) button in the [Formatting](#) ribbon group in Edit Mode, or by right-clicking on the oneline diagram background and choosing Edit Screen Layers.

Screen layers provide a method to filter objects on a oneline diagram based on either zoom level or to which layer the objects are assigned. Initially each object is in a common layer called Default, which cannot be modified or deleted. You can create new layers using the Screen Layers display.

Since the Screen Layers display is a [Case Information Display](#), it shares the of other case information displays. You can access many case information display features by right-clicking on the display to invoke the local popup menu. Most importantly, this menu contains the options to Insert and Delete screen layers.

To delete a screen layer, right-click on that layer in the display and select Delete from the popup menu. The one exception is the Default layer, which cannot be deleted.

To insert a new screen layer, right-click on a record in the display and choose Insert from the popup menu. This will open the [Screen Layer Options](#) dialog for defining the options of the new layer you are inserting. You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Screen Layer > Define Screen Layer**.

Screen Layer Options

The Screen Layer Options are used to define options for new layers, or modify options for existing layers. Devices can be added to a layer by choosing the layer name from the drop-down list on the [Levels/Layers Tab](#) of the [Format Multiple Objects](#) dialog.

Name

Layers can have any name desired to describe the layer. The default naming convention is "Layer #".

Show Layer

If this box is checked, any objects on the diagram which are contained in this layer *may* be visible. This depends on the settings for low and high zoom level, described below. If the box is unchecked, however, then any objects contained in this layer will be hidden. Only objects that are visible will be used when calculating contours. Unlike [Custom Display Detail](#), layers can be applied in both edit mode and run mode.

Selectable in Edit Mode

By unchecking this box the user prevents the objects contained in this layer of being with the mouse during Edit Mode. This behavior is particularly desirable for elements (such as borders which are pretty much already fixed) when editing oneline diagrams.

Use Conditional Display by Zoom Level

Objects can be displayed or hidden based on the Screen Layer to which they are assigned and the current zoom level of the oneline diagram. Checking this box, and setting the Low Zoom Level and High Zoom Level fields, dictates at what zoom range the objects contained in the zoom level will be visible. If the zoom level is outside the range defined, any objects belonging to the screen layer will be hidden. This also applies to both edit mode and run mode.

Memo

This tab allows the user to enter text in order to describe this layer.

Format Field Properties

The Field tab of the [Format Multiple Objects](#) dialog can be used to modify the format of the selected fields, or to change a set of selected fields from one field representation to another. Use the [Select by Criteria](#) dialog to select fields of a particular type, right-click on any of the selected fields and choose **Format Selection** from the popup menu.

Note that this option also applies to objects which display information related to specific object fields, such as [Line](#) and [Interface](#) Pie Charts.

Object Type

This field will be filled in automatically based on the type of object the selected field(s) apply to. You cannot change this field, which means that you can only change the selected field(s) to another field for the same type of object.

Field

The field type of the selected field(s). To change the selected fields to a new field type, click on the drop-down arrow of the combo box, or click on the **Find...** button, and select a new field from the list of fields for the Object Type.

Include Suffix

Checking this option will add the unit suffix to the selected fields.

Total Digits in Field

Modify the number of digits to display in the field. This number includes the decimal and the numbers to the right of the decimal.

Digits to Right of Decimal

Modify the number of the total digits that should appear to the right of the decimal.

Display/Size Properties

Use the Display/Size tab of the [Format Multiple Objects](#) dialog to change the display size, orientation, anchor property of the selected object(s).

Size

This field will change the characteristic display size of the selected object(s). The characteristic size is normally the vertical size or height of display objects. In objects with different orientation choices, the size is the most distinctive dimension of the object (e.g. in buses with right or left orientations, the size is the horizontal dimension of the bus bar, but in buses with up or down orientations, the size is the vertical dimension of the bus bar).

Width

This field will change the characteristic display width of the selected object(s). Similarly to the size, the width represents the horizontal dimension or width of display objects. In objects with different orientation choices, the width is the least distinctive dimension of the object (e.g. in buses with right or left orientations, the width is the vertical dimension of the bus bar, but in buses with up or down orientations, the width is the horizontal dimension of the bus bar).

Anchored

To toggle whether or not the selected object(s) are [anchored](#), check or uncheck this option

Immobile

To force the selected object(s) to stay in the same position, check this option.

Orientation

This group will modify the orientation setting of the selected object(s). The possible settings will change depending on the type of object selected. Some object types do not have an orientation.

Shape

Use this group to modify the shape of the selected display objects. Possible shape settings will change depending on the type of objects selected. If objects of different types are selected and these types do not have a common set of shapes possible, the shape option will be blank and the shape cannot be changed.

For Substation objects, the Use Substation Layout Settings specifies that the shape selected in the [Substation Display Options](#) will be used.

When generator objects are selected, the shape selection determines the shape of the rotor inside the generator. The selections include images that represent possible generator fuel types including: coal, hydro, natural gas, nuclear, oil, solar, and wind. The other images include a dog bone, sine wave, or no rotor image. If selecting a rotor shape that can be filled, **Line Color 2** on the [Line/Fill tab](#) determines the fill color for the inside of the shape.

Number of Selected Objects

This field shows the number of objects that are currently selected on the online.

Align Group Objects

The Align Group feature allows for selecting multiple objects on a oneline diagram and aligning them in a specified manner. There are a couple of ways to perform the alignment.

First, select the objects on the diagram that you wish to align. Once you have selected all objects, you can 1) right click on a selected object and choose **Align Group Objects** from the local popup menu or 2) use the **Alignment** menu found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

The **Align Group Objects** option from the local menu will open the Align Dialog, which allows for specifying the type of alignment desired. Alignment can be performed horizontally, vertically, or both. In addition to alignment of the top, bottom, left and right edges of the selected objects, you can also choose to have the objects snap to the [drawing grid](#) in the horizontal and/or vertical directions as well.

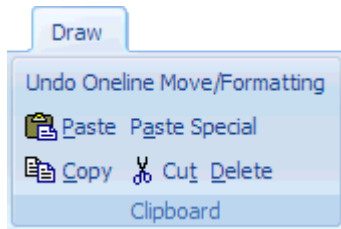
These alignment actions can also be selected directly from the **Alignment** menu found on the Formatting ribbon group on the Draw ribbon tab.

Refresh Anchors

The **Refresh Anchors** option of the [Formatting](#) ribbon group on the [Draw](#) ribbon tab allows you to update or reset the anchoring of objects on a oneline diagram to their respective anchor. It can be advantageous to use the Refresh Anchors option when opening onelines that were created by another case, copying and pasting data between onelines, or renumbering objects on the oneline. This ensures the anchoring of objects to the appropriate anchor on the oneline diagram.

Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



Undo

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the oneline. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

Absolute Coordinates : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different oneline diagrams which contain the same geographic background.

Coordinates Relative to Cursor : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

Undo Command

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

Online Diagram Overview

The purpose of the online diagram is to show information about the power system graphically. Such displays are called online diagrams (onelines) because the actual three-phase power system components are represented using a single line. Simulator onelines "come alive" via:

- [Animation](#)
- [Contouring](#)
- [Zooming and panning](#) capability and
- [Conditional display of objects](#)

Additionally, a key aspect of Simulator is the ease with which it allows the user to examine and modify many of the shown on the online diagram.

The user may open any number of online diagrams, including multiple copies of the same online.

Relationship Between Display Objects and the Power System Model

Display Object:

An item shown on a oneline diagram. Display objects typically have an associated model object. Examples include buses, transmission lines, transformers, generators and loads. Display objects not associated with a model object are called unlinked objects.

Model Object:

A power system element contained in a case.

A key strength of the Simulator is its ability to allow users to manipulate a power system model graphically. This capability greatly simplifies the work involved in developing or maintaining a power system case for both novice and advanced users. However, it is important to keep in mind the distinction between the shown on the onelines and the actual power system model, consisting of model objects. A key concept is that any number of display objects, including none at all, can be associated with a single model element.

Simulator uses a bus-oriented model. In other words, the model objects are either the buses themselves, objects that are radially attached to a bus (i.e., loads, generators and switched shunts), or objects that join two buses (i.e., transmission lines, transformers or dc lines). As long as there is a one-to-one mapping between display objects and model objects, the distinction between the two could be made entirely transparent to the user.

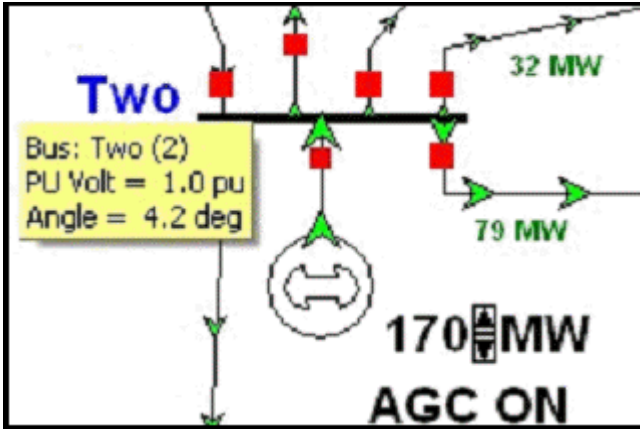
It is reasonable, and often quite useful, to use more than one display object to represent a single model object. For example, by using the [Conditional display of objects](#) feature, two bus display objects could be used on a single oneline to represent the same bus. One bus might be visible over a particular zoom range, while another, with perhaps a different size/thickness, is visible over another range. Alternatively, the same bus could be represented using display objects drawn on separate onelines.

An ambiguity arises when the user uses the Cut command to delete an object. Is he or she deleting just the Display Object or both the Display Object and the Model Object? To alleviate the problem, Simulator prompts you when you are deleting a display object with an associated model object to delete both the Display Object and its associated model object record, delete just the display object, or cancel the delete.

In addition, there is no requirement that model objects have a corresponding display object. Thus, you could use the oneline diagram to show just a fraction of the total system buses and other devices. You can use the [Case Information](#) menu to view the model objects directly regardless of whether or not they are shown on a oneline.

Custom Hint Values

Simulator can be configured to show a short hint string when the mouse is hovered over objects on the oneline diagrams. This hint string will appear in a yellow box as shown below.



This option must be set first by opening the [Simulator Options dialog](#) and then going to the **Online** category and finally checking the **Show Online Hints**. The user may then customize what text and values appear in the hint by choosing **Custom Hint Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These **General Options** are also available on the [Online](#) ribbon tab.

This brings up a dialog on which customizations are made for each type of display object that can show a hint.

The dialog box is titled 'Positions for Displaying Hints'. It has a left pane with a list of display objects: Background Object, Display Area, Display Bus, Display Generator, Display Interface, Display Load, Display Multi-Section Transmission Line, Display Substation, Display Super Area, Display Switched Shunt, Display Transmission Line/Display Transformer, and Display Zone. The 'Display Bus' object is selected. The main pane is titled 'Display Bus Hint Positions' and contains a table with columns 'Field' and 'Example'. The table has three rows: 'Object Identifier' with example 'Identifier for the object appears here', 'Voltage\Per Unit Magnitude' with example 'PU Volt = 123.1 units', and 'Voltage\Angle (degrees)' with example 'Angle = 123.1 units'. There are buttons for 'Up', 'Down', 'Add', 'Remove', 'Modify', and 'Remove ID'. The bottom section has input fields for 'Delay before showing hints (ms)' (300), 'Delay before hiding hints (ms)' (6000), and 'Maximum hint width (pixels)' (200). It also has radio buttons for 'Show Object Identifier with the following Identifier Type': 'Name (Number)' (selected), 'Name', 'Number (Name)', and 'Number'. There are buttons for 'Set Hint Font', 'Recommended Hint Defaults', 'Set Font to Windows Default', and 'Remove All Hints'. At the bottom are buttons for 'Close', 'Help', 'Save All To Aux File', and 'Load All From Aux File'.

Field	Example
Object Identifier	Identifier for the object appears here
Voltage\Per Unit Magnitude	PU Volt = 123.1 units
Voltage\Angle (degrees)	Angle = 123.1 units

Choose the type of object on the left side of the dialog. With this display object chosen, choose which fields of this dialog you would like to include as part of the custom hint. Click the **Add**, **Remove**, or **Modify** buttons to add, remove, or modify a field to the hint. Click the **Up** and **Down** buttons to change the order in which the fields appear. On the dialog that appears after clicking Add or Modify, you may customize the field with a prefix, modify the digits and decimal points, and specify whether to show the units as a suffix. The Example column shows an example of what text will appear for the hint.

The *Object Identifier* is also available that will appear as the first line of the hint if chosen. To remove this identifier, click the **Remove ID** button. When the object identifier is not being shown the **Remove ID** button caption will change to **Add ID**. Click the **Add ID** button to add the object identifier back to the hint. You may specify the format of the object identifier by setting the **Show Object Identifier with the following Identifier Type** option appropriately.

The **Delay before showing hints (ms)** is to set the milliseconds time that the hint will show up after you hover over an object and the **Delay before hiding hints (ms)** is the milliseconds time before the hints are going to be hide. The **Maximum hint width (pixels)** is the maximum pixels width that the yellow box of the hint will have.

By selecting the **Set Hint Font** a dialog will open to select the font type, font style and size of the text in the hint box. Selecting **Set Font to Window Default** will set the text fonts to the Window defaults values.

The hint customizations are stored with the PowerWorld Binary file (PWB) and in the Windows Registry. The customizations affect all oneline diagrams (including the Bus View and Substation View). The customizations may also be stored in an [Auxiliary File](#) for use in moving the customizations between computers and cases. Click the **Save All to Aux File** button to save all these customizations to an Auxiliary File or click the **Load All From Aux File** to load all the custom hints from an Auxiliary File.

Online Local Menu

The local menu provides access to a number of options and tools directly from the oneline. To display the local menu, position the cursor on an empty portion of the oneline then click the right mouse button. While most options are always available on the popup menu, a few options are only available on the popup menu in run mode.

Find Object on Oneline (Edit and Run Modes)

Displays the Find Object on Oneline tab of the [Zoom, Pan, and Find Objects](#) dialog.

Online Display Options (Edit and Run Modes)

Displays the [Online Display Options](#) dialog. This dialog allows you to customize the appearance of the oneline.

Pan/Zoom Control (Edit and Run Modes)

Displays the Zoom/Pan tab of the [Zoom, Pan, and Find Objects](#) dialog.

Edit Screen Layers (Edit and Run Modes)

Opens the [Screen Layers case information display](#).

Show Layer (Edit and Run Modes)

Allows activation of any saved screen layer.

Save/Edit/Delete View (Edit and Run Modes)

Displays the [Save View Level Dialog](#). This dialog is used to set defined locations on the oneline for recalling specific views from a list of saved views.

Go To View (Edit and Run Modes)

Allows the user to go to a specific location and zoom level on the oneline by choosing from a list of saved views. This option does nothing if no views are saved.

Contouring (Run Mode Only)

Displays the [Contour Options Dialog](#). This dialog allows you to contour the system voltage magnitudes or angles.

Difference Flows

Displays the [Difference Flows Dialog](#). This dialog is used to compare two power system operating points.

Dynamic Formatting (Active Oneline) (Edit and Run Modes in Oneline Diagrams)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in oneline diagrams according to the state of the represented object in the power system.

Dynamic Formatting (All Views) (Edit and Run Modes in Bus and Substation Views)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in bus and substation views according to the state of the represented object in the power system.

All Display Objects (Edit and Run Modes)

Opens the [Display Object case information display](#), showing all the objects contained in the oneline diagram.

Only Selected Display Objects (Edit Mode Only)

Opens the [Display Object case information display](#), showing only the currently selected objects in the online diagram, by automatically checking the **Show Only Objects Selected on Display** box.

Load Display Auxiliary File (Edit and Run Modes)

Loads a [display auxiliary file](#).

Print Online (Edit and Run Modes)

Sends a copy of the online diagram to the printer. Selecting this option has the same affect as selecting **File, Print Online** from the main menu. See [Printing Online Diagrams](#) for more information.

Copy Image to Clipboard (Edit and Run Modes)

Copies the online file into the Windows clipboard using the Windows Metafile format (*.wmf). You can then paste the online into another program, such as a word processor. See [Copying Onelines to Other Programs](#) for details.

Export Image to File (Edit and Run Modes)

Saves a copy of the entire online diagram image to a file. Online image files can be saved in bitmap (*.BMP), Windows Metafile (*.WMF, *.EMF), JPEG (*.jpg), or GIF format (*.gif).

Toggle Full Screen (Run Mode Only)

Sets the online diagram display window to full screen size. This option hides all toolbars and status bars. To return the window to normal, right-click on the online diagram and un-check the Full Screen option.

Embed a Display (Run Mode Only)

Clicking on this option allows you to open another online diagram (or the same one) and embed the new display inside the existing display. You can choose what percentage size the embedded display should be, and where within the existing window the embedded window should be placed.

Form Control > (Edit and Run Modes)

Options on this submenu allow you to resize the window, shift the window, or close the window.

Online Display Options Dialog

The Online Display Options dialog allows you to customize the display of the online diagram. You can access this display by right-clicking anywhere on an empty portion of a online and selecting **Online Display Options** from the resulting [online local menu](#).

This dialog can also be display by going to the [Options](#) ribbon tab and choosing **Online Display Options** from the **Online Options** ribbon group.

This dialog houses pages that govern various aspects of the online display. See [Animated Flows](#), [Display Object Options](#), [Display Options](#), [Grid/Highlight Unlinked Objects](#), [Geography/Coordinates](#), [Memo](#), [Pie Charts / Gauges](#), [Thumbnail View](#), or [Substations](#) for information on the respective tabs. Older versions of Simulator used to support a movie maker tool useful with the onelines. See the [Movie Maker](#) topic for suggestions on what tools to use instead.

Save Options to Case

Of important note is the general option at the bottom of this dialog labeled **Save Options to Case**. What this allows you to do is define a set of online options in the dialog, and then save that definition of options *in the case* by giving the definition a name. Once you have named that set of options, you can recall it later by coming back to this dialog and choosing the name from the list of option set names that appears near the bottom of the dialog. You can define as many different sets of custom options as you wish. You can also use an option set saved with the case when you create a online view using the [Save Views](#) dialog.

Also note that once you have defined these sets of Saved Online Options, you can then quickly apply them to the active online diagram by using the **Saved Options Menu** on the [Options](#) ribbon tab.

Display Options

The Display Options page is found on the [Online Display Options](#) dialog. This page contains the following option settings:

Display Detail

The Display Detail box allows you to control how much information is shown on the oneline display. This option is available only in run mode. All elements on a oneline diagram are automatically visible in edit mode. There are four choices for run mode detail filtering:

Minimal Show the oneline background, branch circuit breakers, generator MW output, and load MW/MVR.

Moderate Show all Minimal information, along with bus voltages and all line flow pie charts.

Complete Show all information.

Custom Selecting this option and then clicking the **Set...** button opens the Custom Detail Dialog, which allows you to customize the oneline diagram to hide objects that do not meet your desired specifications. This dialog looks and works very similar to the [Select By Criteria](#) tool, allowing you to choose objects filtered by area, zone, voltage, and screen layer.

Of course, in order for a certain display object to appear, it must have been placed there by the person who designed the oneline.

Emphasize Specific Objects

This option allows you to choose to "emphasize" desired elements on a oneline diagram. The emphasis is in the form of the emphasized elements being in full color, with de-emphasized elements being muted colors.

To choose emphasis of certain elements, first check the box labeled **Do Emphasis**. When you first check this box, it will open the Emphasis Filter dialog, which looks and works very similar to the [Select By Criteria](#) tool. You can choose the oneline elements you wish to emphasize, and include filtering by area, zone, voltage, and screen layer.

The degree of emphasis (muting of de-emphasized elements) can be controlled by the **Emphasis Amount** slide bar.

Background Color

Select **Change Background Color** to select a different color for the oneline diagram background color. Select the **Set as Default Background Color** option to set the background color as the default for all oneline diagrams.

Use Absolute Values for MW Line Flows

If checked, this option will cause all MW flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the line and negative near the sink end of the line. The value of this option is stored in the system registry, and will be used by bus view and substation view diagrams when a oneline diagram with saved oneline options is not available.

Use Absolute Values for Mvar Line Flows

If checked, this option will cause all Mvar flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the Mvar flow text fields will be positive near the source end of the line and negative near the sink end of the line. The value of this option is stored in the system registry, and will be used by bus view and substation view diagrams when a oneline diagram with saved oneline options is not available.

Use Absolute Values for MW Interface Flows

If checked, this option will cause all MW flow text fields for interfaces to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the interface and negative near the sink end of the interface. The value of this option is stored in the system registry, and will be used by bus view and substation view diagrams when a oneline diagram with saved oneline options is not available.

Enable Mouse Wheel Zooming

When this box is checked, zooming can be done with a mouse wheel. The default is off.

This option applies to all onelines instead of just the presently selected oneline.

Show Oneline Hints

When this box is checked, hovering the cursor over an object will briefly display information about the object in a popup hint.

This option applies to all onelines instead of just the presently selected oneline.

Visualizing Out-of-Service Elements

Out-of-service elements can have optional visualization settings that makes them "stand out" in relation to in-service devices on the oneline diagram. The options for out-of-service devices includes **Use Dashed Lines**, **Draw and X Through Off-Line Generators**, and **Blink**. Additional controls for the Blink option are available, for setting the blink interval and color. Any combination of these options can be used for out-of-service elements, although the **X** option only applies to out-of-service generators.

Browsing Path for Oneline Diagrams

This option applies when you have [Oneline Links](#) included on a oneline diagram. Rather than specify the full path and name of a oneline diagram as a oneline link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the oneline file name stored with the link.

Pie Chart/Gauge Options

Show Pie Charts/Gauges in Run Mode

When this option is checked, the pie charts and line gauges will be visible during Run Mode. Otherwise, they will be visible only during Edit Mode.

Only Show Pie Charts/Gauges if relevant data exists

The relevancy of pie charts and gauges depends on the style of the pie chart or gauge. For MVA, MW, Mvar and Amp styles, the pie charts are considered irrelevant if the limits on the line or interface are 0. For max percent loading under contingency, the pie chart is irrelevant if no violation(s) under contingency occurred on the element. For the PTDF style, the pie chart is invalid if the PTDF value has not been calculated for the line or interface.

The options for pie charts and gauges are split into four sections, discussed in the additional topics:

[Pie Charts/Gauges: Lines](#)

[Pie Charts/Gauges: Interfaces](#)

[Pie Charts/Gauges: Pie Chart/Gauge Styles](#)

[Pie Charts/Gauges: General Options](#)

Many of these same options are available on the Pie Chart menu found on the [Options ribbon tab](#).

Pie Chart ▾

	Flow Viz Type	Actual MW ▾
	Pie Chart Style	Total power ▾
<input checked="" type="checkbox"/>	Use Limiting Flow	
	Show Value	80% ▴ ▾
	Normal Scalar	1 ▴ ▾
	Normal	Blue ▾ ...
	1st First Warning	80% ▴ ▾
	1st Warning Scalar	1.5 ▴ ▾
	1st Warning	(custom) ▾ ...
	2nd Warning	100% ▴ ▾
	2nd Warning Scalar	2 ▴ ▾
	2nd Warning	Red ▾ ...
<input checked="" type="checkbox"/>	Limit Set Header	
<input checked="" type="checkbox"/>	Style Footer	
	Resize Only Monitored	
	Max Zoom Resize	500 ▴ ▾
	Pie Font Size	50% ▴ ▾
	Background Method	Online Color ▾
	Specific Background	(custom ▾ ...
	Color as Anchor	

Animated Flows Options

The fields on the Animated Flows page of the [Online Display Options](#) dialog are used to customize the appearance of the animated flows on the oneline diagram.

Show Flow Symbols

Determines whether power flows are animated on the onelines. If this option is not checked, then no flow symbols appear on the oneline.

Show Flows On

Allows you to choose whether or not to display animated flows individually on Branch, Load, Generator and Shunt objects. Uncheck the box(es) for the objects that you do not wish to display animated flows.

Base Flow Scaling on

Animated flows on onelines may depict either actual power flows or [power transfer distribution factors](#), depending on what you choose here. You can also choose to animate both the MW and MVAR flows or the MW and PTDF flows simultaneously. The Custom Float 1 option allows user specification of a value to depict in the flow animation. To use this option, values should be entered in the Custom Floating Point 1 field for the appropriate devices.

Animate

This option determines whether or not the flow arrows are mobile or stationary on the oneline diagram. Unchecking this box will cause the flow arrows to remain stationary on the diagram.

Scale Speed of Flow

Checking this box will cause the arrows to flow at a speed proportional to the represented quantity. When it is not checked, all of the flows will be at the same speed.

Scale Size of Flow

If checked, the size of the animation symbols will vary to represent the magnitude of flow on the element (respective of the **Animation Parameter**.) Otherwise, the symbol size will be the same on all devices regardless of the magnitude represented.

Reset Animated Flow Offsets

This button allows the user to reset the animated flows to start at a specified offset position. Mostly this would be used to reset the offset to 0, which would cause the animated flows to start at the beginning of the line or element. Since the animation moves the flow arrows every time the load flow is resolved, resetting flows to a specific offset could be useful for comparing different load flow solutions by looking at differences in the animated flow objects.

Set Size, Density, and Reference Values for this oneline

Simulator detects the current zoom level and object parameters of the selected oneline diagram, and automatically adjusts the animation settings in an attempt to optimize the animation quality.

Size

Determines the relative size of the animated flows on the devices. Increasing this number increases the size of the flow symbols. This field may range from 1 to 999.

Density

Determines the relative density of the animated flows on the devices. Increasing this value causes Simulator to display a greater number of flow symbols per unit distance on the oneline. This value may range from 1 to 999.

Scaling Based on

Determines whether the size and speed of animated flows represent actual flow or percentage loading. This option applies only to transmission lines and transformers as flows on other devices, such as loads and generators, always represent actual flow.

Max. Zoom Level to Scale Size

As a oneline is zoomed, the animated flow symbols increase in size. The value of Max. Zoom Level to Scale Size caps the size of the animated flows so that zooming beyond this level results in no further increase to the size of the flow symbols.

Maintain Density above Maximum Zoom Level

As a oneline is zoomed, the animated flow symbols decrease in density. Checking this box maintains the density of the animated flows after the maximum zoom level has been reached.

Minimum Size in Pixels for In-service Elements

Allows the specification of a minimum size for the animated flow objects for power systems elements that are in-service. The flow objects are sized based on the flow through the element. For elements with very small flows, it is often difficult to see the flow objects, and the minimum size can be set to make these flow objects more visible. The larger the minimum size value becomes, the more uniform the flow objects will become as elements are no longer sized based on their actual flow but rather the minimum size.

Reference Values for Scaling

Maximum flow reference for sizing the animated flows. The lower the MVA reference, the larger the animated flows appear on the oneline diagram as the actual flow value is compared to the reference value for scaling.

Symbol Shape

Simulator can display animated flows using directional arrows, circles, squares, or triangles.

Animation Rate

If the oneline animation is too fast or too slow, the animation rate can be adjusted by moving the slide bar.

Symbol Fill Color

Shows the fill color used for the animated flows if **Use Fill Color** is checked. Double-click on these fields to change the colors of the six different types of animated flows.

Use Fill Color

Checked if animated flow symbols should be filled using the **Fill Color**.

Show PTDF Counter Flow

Check this option if you wish for PTDF values that are in the opposite direction of the actual flow to be displayed using a different symbol color when visualizing PTDFs.

Many of these same options are available on the Animation Menu found on the [Options ribbon tab](#).

Animation ▾

✓

Show

Density 10

Size 20

Parameter Actual ▾

✓

Animated Flow

✓

Animate Size

✓

Scale Speed

Max Line 200

Max Zoom Resize 200

Minimum Size in Pixels 0

Auto Set

Flow Viz Type Actual MW ▾

MW Fill

Lime

...

Mvar Fill

Blue

...

Fault Fill

Black

...

PTDF Fill

Black

...

PTDF Counter Fill

Black

...

Symbol Arrows ▾

Animation Rate 43

Thumbnail View

The thumbnail view allows the user to see an overview of the online diagram in a smaller window in a specified location on the online diagram. The thumbnail view is useful when the user has zoomed in to a specific area of the online diagram, but still wants to see what part of the system they are observing on a larger scale as they pan around the diagram. The other application is to observe a more detailed part of the system in the thumbnail view as the user moves the cursor over the online diagram. The thumbnail view is not visible by default, but can be set up and displayed from the [Online Display Options](#) dialog. The options for the thumbnail view are:

Show ThumbNail View

If checked, the thumbnail view will be visible in the specified corner of the online diagram.

Size of ThumbNail View

The size of the thumbnail view as a percentage of the size of the online diagram.

Zoom level of Thumbnail View

The amount to multiply the online diagram zoom level for display in the thumbnail view. The higher the multiplier, the more of the diagram you will see in the thumbnail view. You can instead indicate that the thumbnail view should display the diagram at a specified zoom level rather than a dynamically scaled zoom level based on the current diagram's zoom level.

Location

Choose the location on the online diagram in which the thumbnail view is to appear.

Border Width in Pixels

The pixel thickness of the border around the thumbnail view.

Background Color

If **Use One-line Background Color** is checked, the thumbnail background color will be the same as the online diagram. If it is not checked, then a different thumbnail background color can be selected by clicking on the colored box next to the **Use Custom Background Color** label.

Thumbnail view follows mouse cursor

This setting is very useful when is combined with the Zoom Level. Using a low multiplier and checking the Thumbnail view follows mouse cursor box will show a more detailed part of the system over which the cursor is being moved.

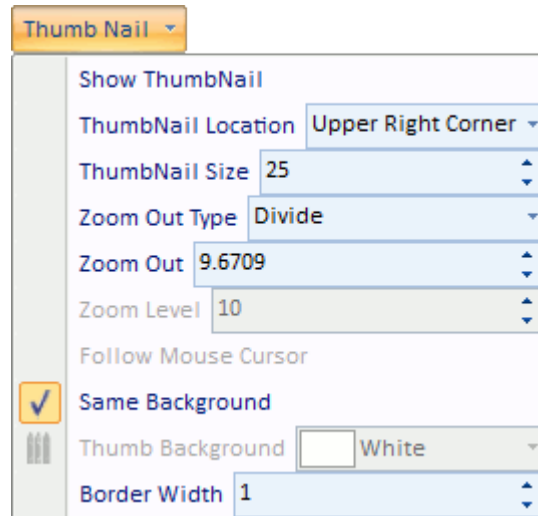
Percent Transparent

This setting allows you to make the thumbnail window completely opaque, completely transparent, or some measure in between. The more transparent the thumbnail window, the more detail of the underlying online diagram is visible through the thumbnail window.

Many of these same options are available on the [Thumb Nail](#) options menu found on the [Options](#) ribbon tab.

Thumb Nail Menu

The Thumbnail View Options quick menu provides access to most of the options contained in the [Thumbnail View Options](#) dialog in an individual manner through menu commands. Select the **Thumb Nail** option from the **Online Options** ribbon group on the [Options](#) ribbon tab to access the menu.



This quick menu contains the following default buttons, in order from top to bottom:

- Show thumbnail on the screen
- Thumbnail location on the screen
- Size of Thumbnail View, as a percentage of the window size
- Zoom out type (divide or zoom level)
- Zoom out factor, defining the ratio of the diagram seen in the thumbnail relative to what is visible in the diagram itself.
- Enable/Disable Thumbnail View to follow mouse cursor
- Enable/Disable Using the Same Background as the main window
- Background Color, if not the same as the main window
- Line width of the border around the thumbnail window

Display Object Options

The Display Object Options page is found on the [Online Display Options](#) dialog. This page contains options for formatting various display objects, displaying circuit breakers on loads, generators, and switched shunts, and formatting circuit breakers for transmission branches.

General Options (All) Tab

Percent of Injection Group/Owner object height used by name

This option allows the user to specify the height of the name with respect to the rectangle of the injection group graphical objects.

Display Circuit Breakers in Generators, Loads and Switched Shunts

Uncheck this box to hide circuit breakers from appearing in generators, loads, and switched shunts. If this option is checked, the user can use **Display breakers on...** to select individually what elements can display circuit breakers.

These options dictate only if circuit breakers will be shown on elements that are closed, i.e. Status = *Closed* for the individual element. Circuit breakers will always be shown for devices that are open, i.e. Status = *Open* for the individual element. This is useful in identifying possible data errors especially for cases where breakers are explicitly modeled. In these cases, the explicitly modeled breakers are used to open and close generators, loads, and switched shunts rather than changing the Status of the device. An open circuit breaker symbol at the individual elements would then indicate that the Status of the device needs to be set to *Closed*.

Transformer Symbol

This option allows the selection of *Coils* or *Circles* to represent a transformer. This option allows a different specification of how transformers should be displayed for each individual oneline diagram. There is also an option that is applied by default for all onelines. Use the *Default* selection to use the setting found with the [Simulator Options: Online](#) options.

Display Rotors in Generators

Uncheck this box to hide the rotors of the generators.

The shape of the rotor can be changed by using the Rotor Shape option found with the [generator display options](#) on the generator dialog in Edit mode or the Shape option found on the [Display/Size tab](#) of the [Format Multiple Objects dialog](#).

Change in Gen Rotor Angle per Refresh (degrees)

This value specifies the change in the rotor angle every time the animated flows refresh. This means that the greater the angle value, the faster the rotor will seem to rotate when the oneline is animated. Entering a negative number will cause the rotor to spin counter-clockwise.

Show Field Inside Generator

Fields can be shown inside generator objects. Select a field from the drop-down box or use the Find button to search through the list of fields available for a generator. Set the field to *No Field* to not display any field inside generator objects. Set the **Digits** and **Decimals** for floating point fields. Adjust the **Relative Font Size** to change the size of the font relative to the generator object size. To ensure that the field fits entirely inside each generator object, check the option to **Dynamically decrease the font size so that the field fits inside the generator**.

Multi-Section Line Display Objects

Intermediate Bus Rotation Angle

Normally, the intermediate buses are perpendicular to the line. The value entered here can rotate the intermediate bus with respect to the original position. Therefore, a value of 0 degrees, will keep the intermediate buses perpendicular to the line, while a value of 90 degrees will set the intermediate buses aligned to the line.

Intermediate Bus Relative Size

This option is actually a multiplier for the size of the intermediate buses specified for each multi-section display object.

Bus Field Voltage Option

This option allows the user to specify in which units the bus fields with *Bus Voltage* as type of field will be displayed. This setting is only valid when the type of field selected is *Bus Voltage* in the [Bus Field Information](#) dialog. Choosing a voltage field in the Select Field will return the voltage value in the specific units of the field.

Circuit Breakers Tab

Circuit Breaker Display Objects > Normally Closed

This section is used to specify the colors for all the circuit breakers normally closed in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

Circuit Breaker Display Objects > Normally Open

This section is used to specify the colors for all the circuit breakers normally open in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

Shape

Specify the shape of circuit breakers on transmission lines and transformers. The breakers shown on loads, generators, and switched shunts will always be the default rectangle shape.

Substation Display Options

The **Substations** tab of the [Online Display Options](#) dialog provides parameters for customizing the appearance of [Substation](#) display objects. This tab is only available if the case contains at least one substation.

The upper right-hand corner of the Substation page displays a template of the substation object's appearance in terms of locations or "zones" where information can be displayed on the object. The rest of the page can be used to describe the appearance of the object and the information that appears in the substation objects.

Upper % of Height

This percentage indicates how much of the object is populated by the "upper" zone of the object. This includes the *Upper Field*, *Upper Left (UL)* and *Upper Right (UR)*. As you decrease this percentage, the upper zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the upper zone.

Identifier % of Height

This is the percentage of the height of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

Lower % of Height

This is the percentage of the substation object that is occupied by the "lower" zone of the object. This includes the *Lower Field*, *Lower Left (LL)* and *Lower Right (LR)*. Unlike the previous two percentages, this percentage cannot be directly modified. Instead, it is automatically determined based on the settings of the *Upper* and *Identifier* percentages.

Left % of Width

This percentage indicates how much of the object is populated by the "left" zone of the object. This includes the *Upper Left (UL)* and *Lower Left (LL)*. As you decrease this percentage, the left zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the left zone.

Identifier % of Width

This is the percentage of the width of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

Right % of Width

This percentage indicates how much of the object is populated by the "right" zone of the object. This includes the *Upper Right (UR)* and *Lower Right (LR)*. As you decrease this percentage, the right zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the right zone.

Buffer Percent

The buffer width for the height settings provide a buffer zone of the percentage width specified horizontally between the Identifier section and the upper / lower field sections. Similarly, the buffer width for the left and right width fields indicates a percentage buffer vertically between the upper / lower field zones and the four corner zones.

Substation Identifier

Choose how the identifier will be displayed for the substation. Choose to display by name, by number, or by combinations of name and number. The identifier will be displayed in the *Identifier* or "middle" zone of the substation object.

Shape

Select the shape used for substations that are not overriding the [Substation Layout Settings](#).

What should be done when identifier text does not fit inside the width

As the name of this option suggest, you can choose how the text in the identifier section of the object should be modified if resizing of the substation object causes the text to be too large for the modified size.

Extra Substation Fields

These four fields, two of which are represented as the *Upper Field* and *Lower Field* on the template, can be customized to display any substation field of your choosing. Use the drop-down arrow of the appropriate box to choose a field, or use the Find button to pull up a list of fields to choose from. Once the field has been chosen, set the digits and number of decimal places for the field. Also check *Units?* to show the units if the extra fields.

Upper-Left Symbol (UL)

Choose what symbol to display in the upper-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Left location of the object to indicate such a presence. By default, the Upper-Left zone will display the Generator symbol.

Upper-Right Symbol (UR)

Choose what symbol to display in the upper-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Right location of the object to indicate such a presence. By default, the Upper-Right zone will display the Load symbol.

Lower Left Symbol (LL)

Choose what symbol to display in the lower-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Left location of the object to indicate such a presence. By default, the Lower-Left zone will display the number of buses in the substation.

Lower-Right Symbol (LR)

Choose what symbol to display in the lower-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Right location of the object to indicate such a presence. By default, the Lower-Right zone will display the Switched Shunt symbol.

Online Animation

An important feature of PowerWorld Simulator is its support of animated onelines. The use of efficient display algorithms allow animation rates that are typically greater than several times per second, even on large cases and on onelines with a significant number of objects. The extensive use of animation makes the display "come alive" so that system conditions can be ascertained more easily.

In Simulator, animation is started from Run Mode by selecting the **Play** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. In Viewer, animation is started automatically when you load a case.

The animation can be controlled and customized from the [Animated Flows Tab](#) of the [Online Display Options](#) dialog. To access this dialog, select **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab, or right-click on an empty area of the online diagram and select **Online Display Options** from the resulting [local menu](#).

Copying Onelines to Other Programs

The onelines can be easily copied to other programs. This allows you to add PowerWorld onelines to your word processor documents or slide presentations. The simplest way to copy a oneline diagram to another program is to use the Windows Clipboard. This is accomplished as follows:

- In Simulator or Viewer, zoom and/or pan the display to the portion of the oneline diagram you would like to copy.
- Right-click on an empty portion of the oneline to display the [local menu](#).
- Select **Copy Image to Clipboard** menu item. This places a copy of the oneline into the Window's clipboard.
- In the other program, use **Paste** or **Paste Special** to copy the contents of the clipboard into that program. The oneline is pasted into the program using the Metafile format.

In addition, oneline diagrams can also be [saved as image files](#) by right-clicking on an empty portion of the diagram and choosing **Export Image to File**.

Display Explorer

The Display Objects case information display presents data describing each graphical object in the oneline diagram. This display functions in the exact same manner as the [Model Explorer](#) and is thus called the **Display Explorer** instead. The Display Explorer contains [Case Information Displays](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated power system object if it has any. You can also sort the object records by clicking on the heading of the field by which you want to sort. Additionally, when Simulator is in [Edit Mode](#), the toolbar **Records** menu allows you to remove existing objects from the oneline diagram.

To show the Display Objects case information display, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group select **List Display > All Display Objects**.

As mentioned above, the Display Explorer functions identically to the [Model Explorer](#). The only differences in functionality are due to additional options found on the bottom left of the dialog. This offers the following additional features.

Show Only Objects Selected

By checking this box, only the selected display objects currently selected in the oneline diagram will be shown.

How to List Grouped Objects

This specifies how to list grouped objects if there are any in the oneline diagram. The choices are to list individually the components of the group, to list only the groups, or to list both the individual elements and the groups. If there are not grouped objects in the oneline diagram, this control is disabled.

Save Complete Display to AXD

By clicking this button, all the objects that are shown in the case information display will be saved to a [display auxiliary file](#).

By default, the display object records case information display contains the following fields:

Type

The type of display object.

X/Longitude Location, Y/Latitude Location

The x and y coordinates of the object's location.

Layer Name

The layer name to which the display object is assigned.

Layer Shown, Selectable in Edit Mode, Low Zoom Level, High Zoom Level

These fields only display information about the layer to which the display object belongs. Therefore, the values for these fields can only be modified through the [Screen Layer case information dialog](#) or the [Screen Layer dialog](#).

Anchored

This field indicates whether or not the display object is an [anchored object](#). If the display object can not be an anchored object, then the field is empty.

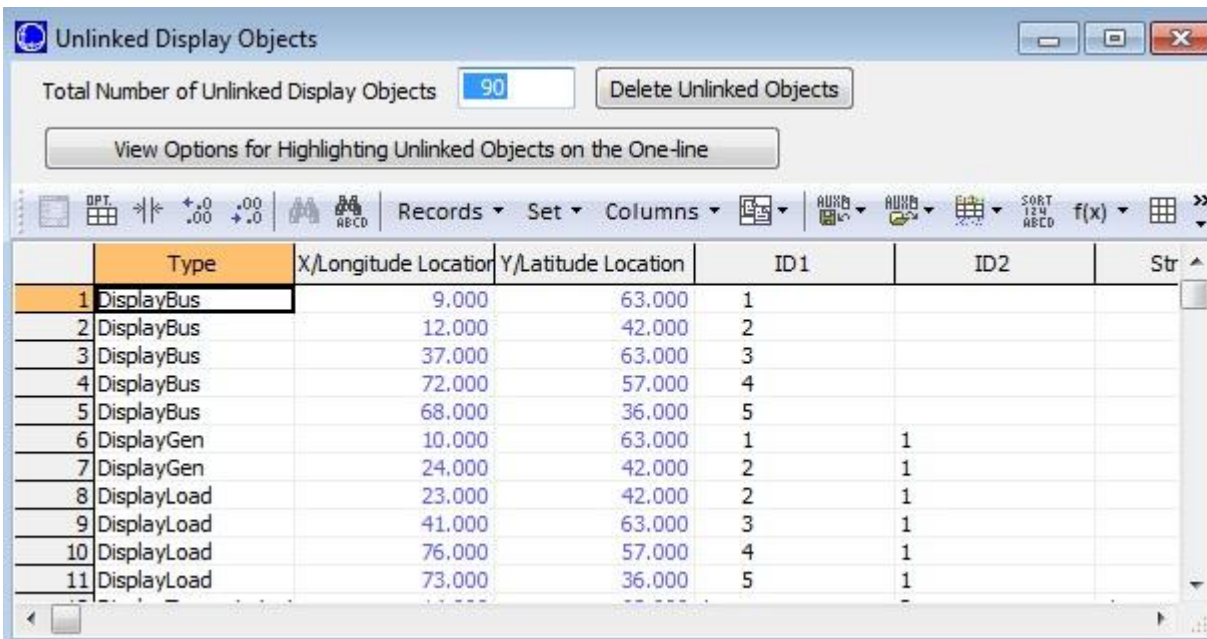
Font Size

For text objects, this field shows the font size.

List Unlinked Display Objects

An unlinked object is a display object not linked to a record in the power system model (model object). The existence of unlinked display objects on a oneline diagram can be misleading because they have zero flows and zero bus voltages associated with them. See [Object Relationships](#) for more information.

Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **List Display > Unlinked Display Objects** from the main menu in Edit Mode to identify unlinked objects.



Unlinked Display Objects Dialog

Total Number of Unlinked Display Objects

Indicates the total number of unlinked objects on the display. Ideally, this number should be zero. If nonzero, the unlinked objects are identified in the table by type, screen location, and zoom range over which the object is visible.

Delete Unlinked Objects

Click on this button to permanently remove all unlinked objects from the display. Exercise this option carefully. Generally, you will want to do this either when you have substantially modified a power flow case, such as by creating an equivalent, or when you are using the oneline with a new power system case.

View Options for Highlighting Unlinked Objects on the Oneline

Click on this button will open the [Grid/Highlight Unlinked Objects](#) tab of the [Oneline Display Options Dialog](#).

Type, X/Y Location

The remainder of the display shows the type, location, identification, layer, applicable zoom level, anchored property, and font size for each unlinked object. This table is a type of [Case Information Display](#) and thus behaves similarly to all other case information displays. Right-click on a record in this table to invoke the local menu. Select **Pan to Object on Open Onelines** to locate and select the unlinked object on the oneline diagram.

All Display Objects

This display works in the exact same manner as the [Model Explorer](#) and is thus called the [Display Explorer](#) instead. This show all of the Display Objects information.

Supplemental Data

Supplemental Data can be used to identify display objects that do not represent power system elements. User defined data types can be created and linked to display objects for use in filtering, dynamic formatting, Select by Criteria, and custom hints.

Supplemental Classification Definition

A Supplemental Classification is used to group Supplemental Data records into categories. As an example, if using Supplemental Data records to identify geographical borders, a Supplemental Classification could be *Country*. Each Supplemental Data record uses exactly one Supplemental Classification record.

Supplemental Classification records are defined by one key field:

Name

This field uniquely identifies the classification and cannot be blank.

Supplemental Data Definition

Supplemental data records are defined by two required key fields:

Classification

This is the Supplemental Classification Name. This field cannot be blank.

Name

This is a unique identifier used to further define the Classification. If using supplemental data records to identify geographical borders, the Name could be *United States* to go with a Classification of *Country*. This field cannot be blank.

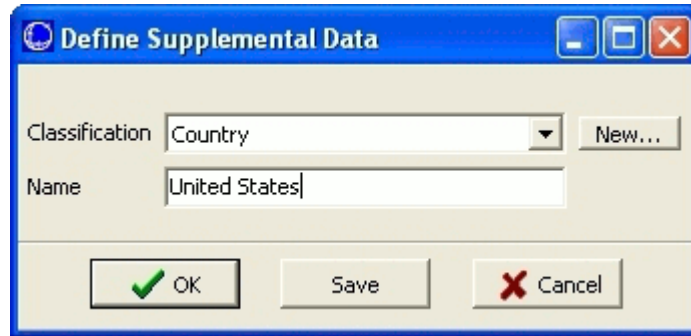
In addition to the Classification and Name, geographic information, expressions, and custom strings, integers, and floating point values can be assigned to supplemental data records. This information can be entered in the Supplemental Data Case Information Display discussed below.

Displaying Supplemental Data Records

Supplemental Data records are displayed in the Supplemental Data Case Information Display found on the [Model Explorer](#) under **Case Information and Auxiliary > Supplemental Data**. This display consists of two tabs, **Supplemental Classification** and **Supplemental Data**. The table on the Supplemental Classification tab lists all of the Classifications that have been defined. The table on the Supplemental Data tab lists all of the Supplemental Data records that have been defined and provides the means for creating new Supplemental Data records AND Supplemental Classification records. Supplemental Classification records cannot be created under the Supplemental Classification tab.

Creating Supplemental Data Records

To create a new Supplemental Data record, from the **Supplemental Data** tab, right-click on the table and choose **Insert** from the local menu. The **Define Supplemental Data** dialog will open.



Either select an existing **Classification** from the drop-down box or create a new Classification by clicking **New**. (If creating a new Classification, another dialog box will open with a prompt for the Classification Name.) Type in a **Name** for the record. To create the record and close the dialog click **OK**. To create the record without closing the dialog click **Save**. To exit the dialog without creating the record click **Cancel**.

Linking Display Objects to Supplemental Data

Display objects that cannot be linked to a power system element can be linked to a Supplemental Data record. These display objects include Background Line, Background Rectangle, Background Ellipse, Background Picture, Text, Case Information Memo, Online Field, Online Link, Document Link, Memo Text, and Generic Model Field.

Linking any of these objects to supplemental data must be done through the **Display Explorer** found under **Onelines > All Display Objects**. From the **Explore** tab, choose the display object type to link. If they are not already displayed, the **Supplemental Data Classification** and **Supplemental Data Name** fields must be added to the list of fields in the table. Simply enter the appropriate Classification and Name in the fields to link a particular display object to a Supplemental Data record.

Once a display object is linked to a supplemental data record, the information in the supplemental data record can be used to filter, dynamically format, Select by Criteria, and display custom hints for that display object.

Because there is no requirement to link display objects that cannot be linked to power system elements to supplemental data records, these objects will not be highlighted along with other unlinked elements when they are not linked to a supplemental data record.

Automatic Linking of Supplemental Data When Inserting Geographic Information

Background objects (lines and ellipses) created from inserting objects from borders included with Simulator and shapefiles can be linked automatically to supplemental data records.

When [auto inserting borders](#) included with Simulator, the option to **Automatically link county and state borders to supplemental data** needs to be checked for this to occur.

When inserting background objects from shapefiles, there are several options that need to be set for proper linking to supplemental data. These options are available on the **Identify** tab of the [GIS Shapefile Data Dialog](#).

Movie Maker

Simulator no longer contains a built-in movie maker tool. Instead, PowerWorld Corporation recommends other movie making tools that can be used from outside of Simulator. Please refer to the PowerWorld website at www.powerworld.com/support/videocapture.asp for references to these third party movie tools.

Map Projections

Background on Map Projections

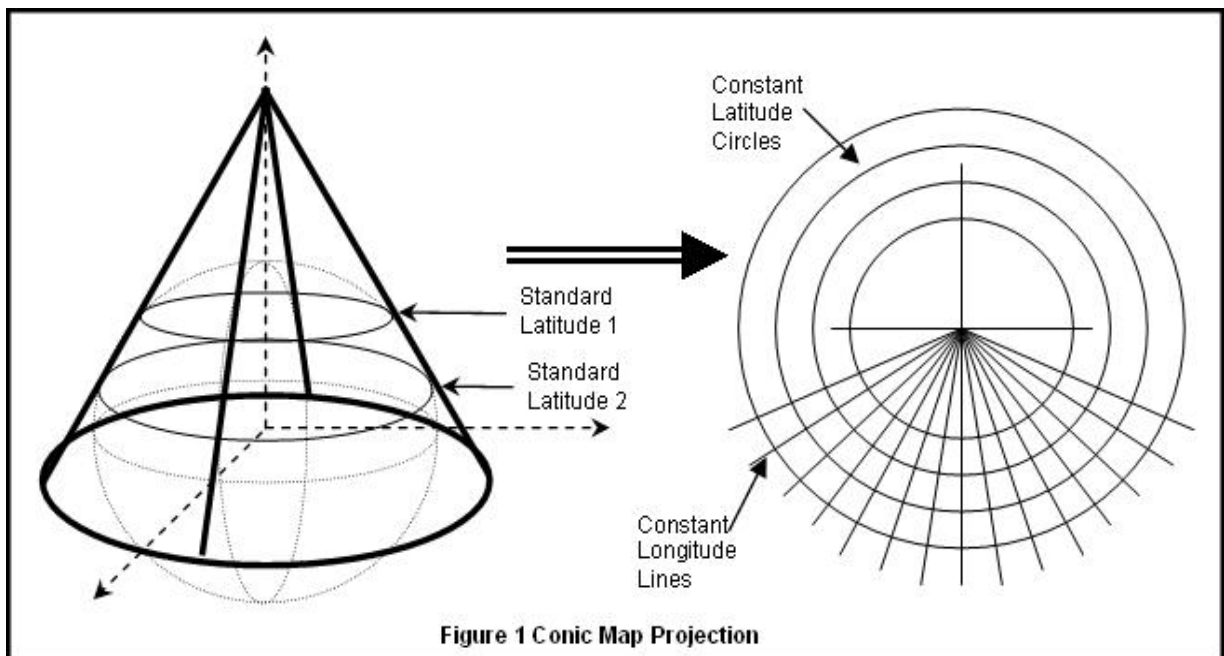
In order to create a map of a spherical object such as the Earth it is first necessary to convert the Earth coordinates of latitude and longitude into a two-dimensional coordinate system needed for a map. This conversion from the spherical coordinates of latitude and longitude to some xy Cartesian coordinate system is called a *map projection*. Mapmakers and mathematicians throughout history have invented many map projections. For each map, the mapmaker must decide which projection will look best for his or her purposes. This will depend on what part of the globe is being drawn, the size of the area being drawn, and the shape of the object being drawn. In general, map projections fall into three standard classes: cylindrical, conical, and planar. For a complete analysis of the topic of map projections see reference 1.

For maps of North America, PowerWorld supports a Simple Conic map projection which is described below.

For maps of the entire Earth, PowerWorld supports a Mercator projection which is described below.

Simple Equidistant Conic Map Projection

For maps made of the 48 conterminous states in the United States, maps are often made using one of the conical map projections. These projections can be recognized by the constant latitude lines that are seen as arcs. When looking at the U.S./Canadian border in the western United States, you will see that it curves on maps made using a conical projection, even though this is a constant latitude border. Several conical projections exist including the Albers Equal-Area Projection, Lambert Conformal Projection, and Simple Equidistant Projection. They can be visualized as taking the globe and placing a cone over it as shown in Figure 1 and then using this to map the coordinates to the xy plane shown at the right. Constant longitude lines are seen as straight lines and constant latitude lines are seen as arcs on the resulting map.



The equations for converting back and forth between (longitude,latitude) and (x,y) using the simple equidistant projection are described next. These equations are taken from reference 1.

First, you must specify seven values that are parameters of the mapping:

- R = Radius of the sphere (earth) in the units used on the map
 ϕ_o = The latitude of the "shifted" origin
 λ_o = The longitude of the "shifted" origin
 ϕ_1 = The first standard parallel
 ϕ_2 = The second standard parallel
 x_f = The shift in the x-direction. Referred to as "false-easting" in some map projections.
 This is done so that all coordinates are positive.
 y_f = The shift in the y-direction. Referred to as "false-northing" in some map projections.
 This is done so that all coordinates are positive.

For these values, the following variables are defined:

$$\rho_o = R(G - \phi_o) \qquad G = \frac{\cos \phi_1}{n} + \phi_1 \qquad n = \frac{\cos \phi_1 - \cos \phi_2}{\phi_2 - \phi_1}$$

Then given a latitude = ϕ and a longitude = λ , calculate the x and y coordinate using

$$\begin{aligned}
 \rho &= R(G - \phi) \\
 \theta &= n(\lambda - \lambda_o) \\
 x &= \rho \sin \theta + x_f \\
 y &= \rho_o - \rho \cos \theta + y_f
 \end{aligned}$$

And given an x and y coordinate, calculate the latitude(ϕ) and longitude(λ) using

$$\begin{aligned}
 \rho &= \pm \sqrt{(x - x_f)^2 + (\rho_o - y + y_f)^2} \\
 &\text{taking the sign of } n \\
 \theta &= \arctan \left[\frac{x - x_f}{\rho_o - y + y_f} \right] \\
 \phi &= G - \frac{\rho}{R} \\
 \lambda &= \lambda_o + \frac{\theta}{n}
 \end{aligned}$$

Figure 2 shows more clearly what some of these variables are.

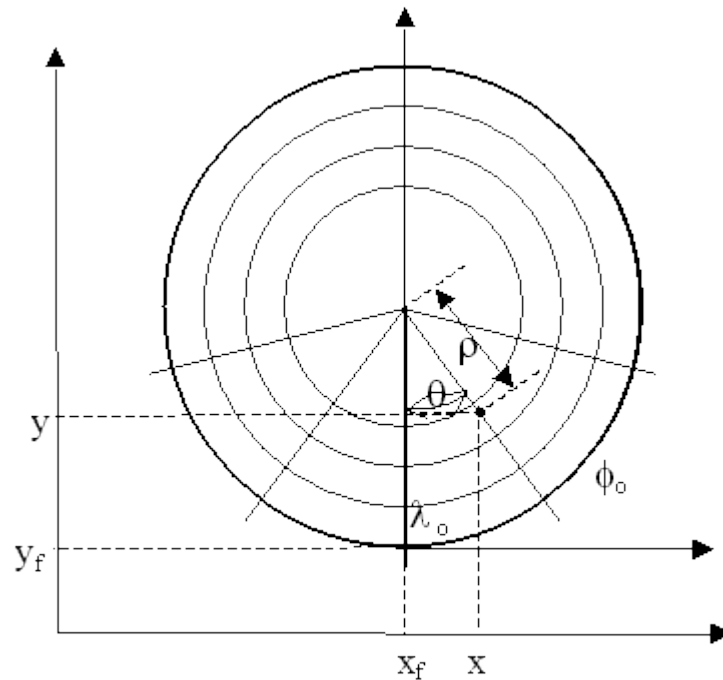


Figure 2 Defining Variables for Conic Projections

Maps made using PowerWorld's built-in map borders for North America use the following parameters:

$$\begin{aligned}
 R &= 30700.0 \\
 \phi_0 &= +9.7 \text{ degrees} \\
 \lambda_0 &= -96.5 \text{ degrees (negative because in Western Hemisphere)} \\
 \phi_1 &= 0.00 \text{ degrees} \\
 \phi_2 &= 50.00 \text{ degrees} \\
 X_f &= 20968 \\
 Y_f &= 0
 \end{aligned}$$

These numbers were thus set to be the default map projection numbers within PowerWorld. This projection will work well for the one-line diagrams with PowerWorld Corporation has created on the same standard background.

Mercator Projection Use in PowerWorld

For the Mercator Projection, PowerWorld assumes $R = 30700.0$ units. From this, given a latitude $= \phi$ and a longitude $= \lambda$, calculate the x and y coordinate using

$$\begin{aligned}
 x &= R * \lambda \\
 y &= R * \ln \left[\tan \left(\frac{\pi}{4} + \frac{\phi}{2} \right) \right]
 \end{aligned}$$

And given an x and y coordinate, we calculate the latitude(ϕ) and longitude(λ) using

$$\lambda = \frac{x}{R}$$

$$\phi = \tan^{-1} \left[\sinh \left(\frac{y}{R} \right) \right]$$

References

1. Snyder, John P., Map Projections - A Working Manual, U. Geological Survey Professional Paper 1395, United States Government Printing Office, Washington, D.C.: 1987.

Great Circle Distance Dialog

To find the Great Circle Distance between any two points of longitude, latitude, select **GIS Tools > Great Circle Distance** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab and the Great Circle Distance Dialog will be displayed. This can be done in Edit or Run Mode.

The Great Circle Distance Dialog allows the calculation of distance between two points of longitude, latitude. The [Great Circle Distance Calculation](#) assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth. The two longitude, latitude points can either be entered manually for any two valid points or can be chosen from valid longitude, latitude values stored with either buses or substations. Valid longitude values are between -180 and 180 degrees. Valid latitude values are between -90 and 90 degrees. The calculated distance is given in either kilometers or miles.

From Location

Click the **Choose From Location** button to choose either the location of a bus or substation as the **From Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the From Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

To Location

Click the **Choose To Location** button to choose either the location of a bus or substation as the **To Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the To Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

Units

Choose the units in which to report the calculated distance.

Calculate Distance

Click this button to calculate the distance in the chosen Units between the From Location and the To Location. The calculated distance will appear in the box above the button. *Error in input data* will be returned if there is a problem with any of the latitude, longitude information.

Great Circle Distance Calculation

The Great Circle Distance Calculation allows the calculation of distance between two points of longitude, latitude. The Great Circle Distance Calculation assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth.

The calculation consists of the following parameters:

- R = Radius of the Earth (6378.137 km from NAD83/WGS84 datum)
- ϕ_0 = The latitude (radians) of the starting location
- λ_0 = The longitude (radians) of the starting location
- ϕ_1 = The latitude (radians) of the ending location
- λ_1 = The longitude (radians) of the ending location
- d = Distance in kilometers between starting and ending location

The distance in kilometers between the starting and ending location is determined from the following equation:

$$d = 2R \arcsin \left(\sqrt{\sin^2 \left(\frac{\phi_0 - \phi_1}{2} \right) + \cos(\phi_0) \cos(\phi_1) \sin^2 \left(\frac{\lambda_0 - \lambda_1}{2} \right)} \right)$$

This calculation is used with several GIS tools to determine the distance between two locations or facilities on a diagram.

Populate Lon,Lat with Display X,Y

To populate the longitude,latitude fields of buses and substations with the locations of corresponding display objects converted to longitude,latitude using the selected projection, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Populate Lon,Lat with Display X,Y**.

The Populate Lon,Lat with Display X,Y Dialog will be displayed. This can be done in either Edit or Run Mode. The dialog will show which map projection is currently in use for the oneline. Users should be careful to not change the map projection once one has been established so that all objects on the oneline are consistently using the same projection. If no projection has been set, the dialog shows **None (x,y)** as the Map Projection, selecting a map projection will establish that map projection as the current oneline map projection.

Map Projection

None (x,y)

No map projection is in use.

North American (simple conic)

This projection is better when dealing with longitude, latitude points in North America. This is the projection that has been used in Simulator for years in defining geographic borders and placing them on a oneline.

Entire World (Mercator)

This projection is better when dealing with longitude, latitude points spread throughout the world.

Click **Populate Longitude,Latitude** to complete the conversion. The longitude, latitude values converted from display x, y coordinates can be viewed on the bus and substation case information displays or the information dialogs.

Geography/Coordinates

The option is available to display screen coordinates in longitude, latitude instead of x,y. This option can be set by selecting **Online Display Options** from the [Onelines](#) ribbon tab or the [Options](#) ribbon tab. The following options are located under the **Geography/Coordinates** category.

[Map Projection](#) in Use

The map projection indicates which projection has been selected for inserting objects on the online. Users should be careful to not change a projection once one has been established. Otherwise, there is the risk that different objects will be inserted with different projections.

Show longitude, latitude coordinates when showing x, y coordinates

When this option is checked and a map projection other than *None (x,y)* is selected, longitude, latitude coordinates will be displayed anywhere that x,y screen coordinates are normally displayed. This option will only be enabled when a map projection other than *None (x,y)* has been selected.

Shape File Import

Simulator allows you to import ESRI shapefiles (*.shp/*.dbf pairs) as a group of background lines or points on a oneline diagram. There are several locations where you may open the GIS Shapefile Import Dialog.

- Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert GIS Data from Shapefile**.
- Go to the [Draw](#) ribbon tab, then on the [Individual Insert](#) ribbon group choose **Background > Insert GIS Data from Shapefile**
- Go to the [Draw](#) ribbon tab, then on the [Quick Insert](#) ribbon group choose **Auto Insert > Insert GIS Data from Shapefile**

The GIS Shapefile Data Dialog has the following sections:

[Control](#)

[Modify](#)

[Identify](#)

[Format](#)

[Shape List](#)

GIS Shapefile Data: Control

This is the first page of the [GIS Shapefile Data](#) dialog. The control page contains the controls for reading a shapefile into Simulator and provides some basic information on the shapes read from the file.

Read in Shapefile

Pressing this button will allow you to choose a shapefile and will load it into Simulator for placement on the currently selected oneline diagram.

Save Shapefile

Clicking this button will save to file the shapefile information currently loaded in memory. This option is useful if making changes to shape format attributes or eliminating certain shapes from those read in and then saving these changes in the shapefile format.

Shapefile Information

Once the shapefile has been loaded into memory, this section will be populated with general information on the number of shapes loaded, and the maximum and minimum X and Y values (usually in longitude and latitude.)

Transfer Shapefile Objects to Oneline

If you are ready to place the shapes on the oneline diagram, but want to continue working with existing shapes or read more shapes from another file, press the **Transfer Shapes to Oneline and Clear** button. This will keep the dialog open following the transfer. It also gives you options of what should happen to the current shapes in memory after you have transferred them to the diagram. You can choose to clear only the shapes that were transferred, all of the shapes, or none of the shapes. The first choice relates to options you have on the [Shapefile Objects](#) page regarding whether or not to transfer certain shapes from the shapefile to the diagram.

Insert Into Layer

Before transferring the shapes to the oneline diagram, you can choose which [layer](#) the shapes should be assigned to. If you wish to place the shapes into a new undefined layer, click on the **Define Layer** button to access the [Screen Layers](#) table.

Transfer Shapes to Oneline and Close

If you wish to transfer the currently loaded shapes to the oneline and close the dialog (clearing the shapes from memory), press this button to complete the process.

GIS Shapefile Data: Identify

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, identification options can be specified on the Identify page.

Identification String for Display Auxiliary Files

Shapefile shapes are created as background objects when added to a diagram. If using [display auxiliary files](#) to create or edit diagrams, objects must be uniquely identified. Because background objects typically do not link to an object in the power flow case, and therefore have no other unique means of identification, a unique ID string must be assigned to objects created via shapefiles. This unique identifier populates the **Auxiliary ID** field used in [display auxiliary files](#) and can be viewed from the [Display Explorer](#).

Prefix

Optionally, a prefix can be concatenated with the selected Attribute to further identify the shape.

Attribute

The drop-down box is populated with the fields that are available with the .dbf file. Choose a field whose value will be used to uniquely identify the shape. By default, the *Record Number* of the shape is used for identification if no other Attribute is selected.

Suffix

Optionally, a suffix can be concatenated with the selected Attribute to further identify the shape.

Linkage to Supplemental Data

Background objects created from shapefiles can be linked to [Supplemental Data](#) records for purposes of further identifying the objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from shapefiles.

Link Display Objects to Supplemental Data

Check this box to link the background objects created from shapefile shapes to Supplemental Data records.

Supplemental Classification

Select an existing Supplemental Classification from the drop-down box or click **New** to create a new Supplemental Classification.

Attribute for Supplemental Data Name

Select an Attribute from the drop-down box. The list of attributes is populated from the fields that are available in the .dbf file. For each unique Attribute value, a supplemental data record will be created with the chosen Supplemental Classification and with its Name set to the Attribute value.

GIS Shapefile Data: Modify

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Modify page can be used to modify the XY attributes of the shapes.

[Map Projection](#)

There are two map projections you can choose from when placing the shapes onto a oneline diagram. The choices are:

- North American (simple conic)
- Entire World (Mercator)

Once you have selected which projection to use, click on the **Convert to Specified Map Projection** button to process the conversion on the shapes currently in memory.

Shift/Scale Shapefile X/Y Data

You also have the option to shift the XY coordinates of shapes or scale them by scalar values. To shift shapes, enter scalar shift values for the X value and Y value and press the **Shift XY Data** button. To scale the XY coordinates, enter scalar scaling factors for the X and Y value, and press the **Scale XY Data** button.

GIS Shapefile Data: Format

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Format page can be used to modify the appearance attributes of the shapes.

Change Shape Format Characteristics

Point Size

The size of points read from the shapefile can be adjusted when transferred to the online diagram.

Line/Border Thickness

Choose the pixel thickness of the shapefile lines when transferred to the online diagram.

Line/Border Color

Set the line color of the shapefile lines when transferred to the online diagram.

Fill Color for Points/Polygons

Set the fill color of the shapefile points and polygons when transferred to the online diagram. The box labeled **Use Fill Color** must be checked as well.

Stack Level

Indicate which [stack level](#) should be applied to the shapes when they are transferred to the online diagram.

Immobile

Check this box if you wish for the shape file objects to be immobile once they are transferred to the online diagram. This will prevent you from inadvertently selecting and moving the shapefile objects on the online diagram while in [Edit mode](#).

Set all shape format attributes to the values above

Once you have finished changing the format settings for the shapes, press this button to apply them for the transfer of shapes to the online diagram.

Automatic Color Mapping

Available Attributes

To customize the shapefile shapes based on a field value read from the .dbf file, select an attribute from the list.

Color Map

Select a color map to apply to the selected attribute. To modify existing or create new color maps, click on the **Modify Color Maps** button. The color of the object will then be based on where its selected attribute value maps to the selected color map.

Change Border Color, Change Fill Color

These two boxes allow you to choose if the color map selected should apply to the border color, the fill color, or both.

Change Color Based on Attribute/Color Map

Once you have finished setting the color map for a selected attribute, press this button to apply the settings before the transfer of shapes to the online diagram.

GIS Shapefile Data: Shape List

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Shape List page can be used to flag shapes for inclusion in the transfer and customize the appearance attributes of specific shapes.

This page will be blank until you have actually read a shapefile into memory. Once you have done so, you will see a list of all shapes read from the file. The table allows you to customize the appearance and attributes of a specific shape, similar to the options on the [Format](#) page.

In addition, you also can choose to include or exclude shapes from the transfer to the online diagram. This gives you the greatest flexibility for hand-picking which shapes you want to add to a online diagram.

Shapefile Database Record Dialog

The Shapefile Database Record dialog is available by selecting **Show Shapefile Fields...** from the local popup menu (obtained by right-clicking on a display object) of any display object that was created from a shapefile. This menu option will not be available for display objects that were not created from a shapefile. This dialog provides information read from the database file (.dbf) associated with the shapefile from which the display object was created. Each time that this dialog is displayed, the database file is read to fill in the field information. The field information is not stored in the PowerWorld display file (.pwd) so the associated database file must be available for this information to be displayed.

File Name

This is the path and name of the database file (.dbf) associated with the shapefile from which the display object was created.

Record Number

This is the record number of the display object within the shapefile.

List of Fields

This list details the fields and values belonging to the given record number in the database file.

Edit Online Browsing Path

Select this option to update the Online Browsing Path. This browsing path will be used to search all listed directories for the database file specified in File Name if it cannot be found in the directory specified with the File Name.

Export Online As Shapefile

Display objects can be exported into shapefile formats (.shp/.dbf pairs) by going to the [Onelines](#) ribbon tab, then in the [Active](#) ribbon group, choosing **GIS Tools > Export Online as Shapefile**. This will open the Select Objects and Fields to Export Dialog, which is similar to the Select by Criteria Dialog.

The dialog is used to select one type of display object to export to the shapefile and the criteria used to select which specific objects will be included in the export.

Selecting Areas and Zones

Use the **Area** and **Zone** fields to select the areas and zones in which to select display objects. [Ranges](#) of area and zone numbers can be entered in the usual way, or the **All Areas** or **All Zone** boxes can be checked to select all areas and all zones respectively. The **Areas** and **Zones** tab pages can also be used to select the areas and zones in which to select display objects.

Selecting Voltage Levels

Specify the max and min voltage levels using the boxes that are provided, keeping in mind that all voltages are in kV. The **All Voltages** box can be checked to include all voltages without requiring the specification of a voltage range.

Selecting Layers

All layers can be selected by choosing the **All** option for Layers. If specific layers are to be selected, the **Range** option should be selected and the **Layers** tab should be used to check and uncheck specific layers in which to select display objects.

Type of Drawing Object

Select the type of object to export in the **Type of Drawing Object** dropdown box. Alternatively, the **Find...** button next to the dropdown can be used to select the object type from a list of available object types.

Filter

The **Filter** dropdown box provides a list of filters available for the selected Type of Drawing Object. The **Find...** button next to this dropdown can be used to find a filter or define a new filter for the selected object type. Filter options will not be enabled when an object type that does not allow filtering is selected.

Selecting Fields

The **Fields** list will be populated for object types that have associated fields. The selected fields will be written to the database file (.dbf) along with the value of the field. When the **Specified** option is selected, individual fields can be chosen by checking the box next to specific fields. When the **All** option is selected, then all fields associated with the selected type of object will be included in the database file. When the **Show Only Commonly Used Fields** option is checked, only those fields that are deemed to be commonly used will be displayed in the list. To show all fields associated with the selected object type, uncheck this box. Use the **Check All** and **Uncheck All** buttons to check all or uncheck all of the fields.

Advanced Selection Criteria

If only objects that are currently visible on the display (due to layering, etc.) are to be selected, check the box labeled **Select only currently visible objects**. If some display objects have been selected prior to opening the dialog, an advanced selection option becomes available. If the box

labeled **Use as a filter on presently selected objects** is checked, the criteria chosen in the dialog will only affect the previously selected objects. In other words, only the objects that were previously selected AND that match the chosen criteria will remain selected when exporting to the shapefile.

Export Coordinates in Longitude, Latitude

If a valid map projection has been selected for displaying display object coordinates in longitude, latitude, then the option to **Export coordinates in longitude, latitude** will be enabled. (A valid map projection can be set with the Map Projection in Use Option found in the [Geography/Coordinates](#) category of the Online Display Options dialog.) By default, the Export coordinates in longitude, latitude option will be checked when it is enabled. When checked, the coordinates of the display objects will be exported to the shapefile in longitude, latitude instead of PowerWorld Simulator x,y coordinates.

Saving Shapefile Export Descriptions

Shapefile Export Description settings can be saved with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This will allow the quick recovery of settings that may have been previously used. Use the **Rename** button to rename an already saved set or the **Delete** button to remove a previously saved set of criteria. If Shapefile Export Description sets have been saved, they can be exported to an auxiliary file by choosing **Save to AUX file**. The **Load from AUX file** button will load an entire auxiliary file regardless of whether it contains Shapefile Export Descriptions or not.

Saving the Shapefile

After all criteria have been set, click **OK**. A dialog will open prompting for the name of the file to save. Enter a file name and click **Save**. The type of file that is specified in the dialog is a shapefile with .shp extension, however, three files will be saved. All files will have the name specified in the save dialog, but the extensions will be .shp, .dbf, and .shx.

Insert Measure Line

Distances between display objects can be approximated by inserting a measure line on the display. A measure line is similar to a background line. It can be drawn as a straight line between two points or as a line with many vertices and segments. By default, a measure line is drawn as a yellow line with a pixel thickness of 4.

To insert a measure line go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert Measure Line**. Then position the mouse cursor on the display where the line should begin and click and release the left mouse button. Move the mouse to the desired termination point of the first line segment. A straight segment will follow the mouse movements. Click and release the left mouse button to complete the line segment and to prepare for drawing the next segment, or double-click if this is the last line segment. To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where the freehand shape begins and drag the mouse to trace the desired shape (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape. At this point, either another freehand section or a straight line segment can be added. When all desired freehand and straight line segments have been added, double-click the left mouse button to complete the line.

To change the shape of the line, first left click on the line to select it. Handles will appear at each vertex. A vertex can be moved by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where the new vertex should be added. Note that freehand lines are nothing more than a continuous series of vertices.

Once the measure line has been added to the display, the length of the line can be displayed by right-clicking on the line and selecting **Measure Length...** from the local menu. The units of the length are dependent on the map projection in use and whether the option to **Show longitude,latitude coordinates when showing x,y coordinates** is checked on the [Online Display Options dialog](#). If the map projection in use is something other than x,y and the option to show longitude,latitude coordinates is checked, the length will be given in miles and kilometers. Otherwise, the length will be given in Simulator units. The [Great Circle Distance calculation](#) is used to determine the distance between vertices on the line to determine the approximate length in miles and kilometers.

Delete All Measure Lines

All [measure lines](#) can be deleted from the display. To do this go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Delete All Measure Lines**.

Path Distances from Bus or Group

To access the Determine Path Distances from Bus or Group dialog go to the [Tools](#) ribbon tab, and choose **Connections > Determine Path Distances to Bus** from the [Other Tools](#) ribbon group. The Determine Path Distances from Bus or Group dialog provides a way to calculate how far electrically every bus in the system is from a particular starting point.

After clicking the **Calculate** button, all buses that are part of the start element will be labeled internally as being a path distance zero. Then a calculation will be done that determines for *each bus* in the system the shortest total path length in getting from each bus to the Start Element. Buses that cannot reach the Start Element will be flagged with a very large distance (by default 10,000). The results of the calculation will then be copied into the Bus Field specified under the Bus Field to Populate option. The results will be displayed in a case information display at the bottom of the dialog.

The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch. Lines to Process

Lines to Process

Regardless of the Distance Measure above, you can choose which branches allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed. Bus Field to Populate

Bus Field to Populate

Choose a field into which the resulting path distances will be copied. It is recommended that you use the custom floating point or integer fields for this purpose.

Closet Facilities to Point

To produce a list of display objects and their distance from a point selected on the one-line diagram, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Closest Facilities to Point**. After selecting this option, the mouse cursor will become a crosshair. Move the cursor to a location on the diagram and left-click. A dialog will open listing all of the display objects and their distance from the selected point in ascending distance from the point.

Select to Show Display Objects of Type

This allows the option of showing all display objects in one list or only listing objects of a particular type.

Distance

This field is listed by default with the display objects. This shows the distance between a given display object and the selected point. The location of the selected point is given by the **X/Longitude** and **Y/Latitude** fields on the dialog. The location of a given object is provided by the **X/Longitude Location** and **Y/Latitude Location** fields with the display object. The units of the distance is given in the Units field on the dialog.

X/Longitude and Y/Latitude

Each display object listed has a corresponding **X/Longitude Location** and **Y/Latitude Location** field. The **X/Longitude** and **Y/Latitude** fields given on the dialog provide the location of the point that was selected on the diagram. The fields on the dialog can also be manually adjusted to determine the distance between the display objects and a given coordinate.

All of the fields that give the location of either the display object or the point selected on the diagram are given in x,y coordinates if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the coordinates will be longitude, latitude. Both of these options can be set from the Online Display Options dialog in the [Geography/Coordinates](#) category.

Units

This indicates the units that the distance is in. The units will be in PowerWorld units if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the units will be in either miles or kilometers. To display the distance in miles, the option to use the **English Measurement System** must be selected on the PowerWorld Simulator Options dialog under the [Environment](#) category. Select **Metric (SI)** as the Measurement System to display the distance in kilometers.

Recalculate Distances

Click this button to recalculate distances after changing the coordinates of the selected point by entering new values for the X/Longitude or Y/Latitude fields.

Online Screen Coordinates

Onelines can be any size and can contain any number of objects. The size and position of objects on the screen are specified in terms of x-y "online screen coordinates." When the zoom level is 100% the size of the online in screen coordinates is 100 by 100. More generally, the size of the online is 100^2 divided by the zoom level in both the x and y directions.

The default screen center is the point (50,50) but this can be easily changed (See [Online Panning and Zooming](#).) Negative screen coordinates are allowed. Usually you will not have to be concerned about an object's location in screen coordinates.

The [Status Bar](#) displays the current cursor screen coordinates while in Edit Mode.

Edit Mode	X = 0.11 Y = 31.33
-----------	--------------------

Status Bar Showing Screen Coordinates

Online Zooming and Panning

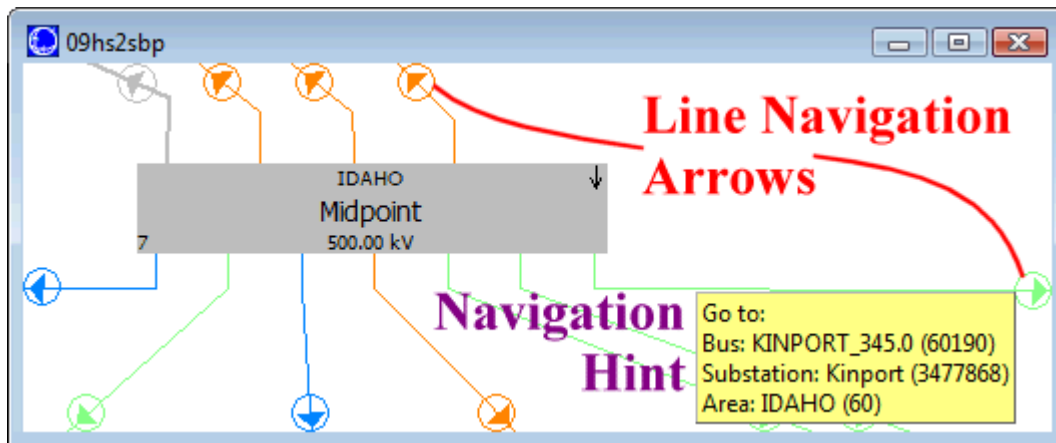
All oneline diagrams permit zooming and panning. Zooming and panning are very important tools for building and investigating large oneline diagrams that occupy more than a single screen of viewing area. Moreover, the display of various objects can be set to depend on the zoom level. See [Online Conditional Display of Objects](#) for more information on this feature.

The following mechanisms are provided for zooming or panning a oneline:

Line Navigation Arrows

For transmission lines whose end-points are not visible at the present pan/zoom level, small Line Navigation Arrows will appear where the line exits the present window. If you click on these arrows, the oneline will automatically pan over to the other end of the transmission line. The oneline will zoom out as it moves away from the present position and zoom back in until it finishes at the same zoom level with the endpoint in the middle of the oneline. On the [Online Tab](#) of the [Simulator Options](#) dialog you can control three attributes for these arrows

1. The size in pixels (default is 9)
2. How fast the program pan/zooms to the new location in seconds. (Making this shorter gets you there faster, but you lose some of the overview information of where you're going as you pan to the new location.)
3. Whether to show hints as you move the mouse over the arrows.



Zooming on Onelines

Zooming involves adjusting the oneline diagram's display area by changing the magnification of the view. Zoom in on a oneline to have the screen display less of the complete oneline diagram, and zoom out on a oneline to have the screen display more of the complete oneline diagram.

To zoom on a oneline diagram using the keyboard, follow these instructions:

- Use CTRL-up arrow to zoom in
- Use CTRL-page up to zoom in quickly
- Use CTRL-down arrow to zoom out
- Use CTRL-page down to zoom out quickly
- The [Zoom](#) ribbon group offers additional zooming options:
- Use the **Zoom in on Area** button of the [Zoom](#) ribbon group to select a region on which to zoom.

- Use the **Show Full** button of the [Zoom](#) ribbon group to zoom the display out to show the entire oneline.
- Use the **Find** button of the [Zoom](#) ribbon group to display the [Pan/Zoom Dialog](#).
- Use the **Save View** button of the [Active](#) ribbon group to display the [Save View Dialog](#), or click on the drop down arrow on the Save View button to choose from a list of saved views.
- Use the **Rectangular Zoom Selector** to select a section of the diagram to zoom into using a selection box. After clicking button (symbolized with a magnifying glass over a dashed rectangle,) left click and hold the mouse button down on the diagram, drag the mouse to select part of the diagram inside a box, then release the mouse button. Simulator will zoom into the region selected inside the box.

Panning on onelines

Panning moves the screen's focus point around the oneline diagram. You can pan left, right, up, or down to view different portions of the complete oneline diagram.

To pan around the oneline using the keyboard, follow these instructions:

- Use the arrow keys to move in the desired direction
- Use page up to move up quickly
- Use page down to move down quickly
- Use the Home key to move left quickly
- Use the End key to move right quickly
- Left-click and hold the mouse button down anywhere on the *background* of the diagram. Make sure you do not click on an object on the diagram. While holding the left mouse button down, drag the mouse in any direction to "drag" the diagram in that direction.

You can also use the [Pan/Zoom Dialog](#) to pan to a specific location, or even to a specific bus. You can display the Pan/Zoom Dialog either by pressing the **Find** button on the Zoom ribbon group of the Onelines ribbon tab or by selecting **Pan/Zoom Control** from the oneline diagram's [local menu](#).

Save View Level Dialog

The **Save View Level Dialog** is used to save an (x,y) location and zoom level, contour description, and/or hidden layer application for a oneline diagram in an easily accessible list for quick recall and application to the diagram. This dialog can be called by selecting the **Save/Edit/Delete View** option from the [online local menu](#).

You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Save View > Save View**.

Recalling a saved view can be done by either right-clicking on the background to access the [online local menu](#) and choosing a view from the **Go To View** list, or by clicking on the drop down arrow on the **Save View** button and selecting a view from the resulting list. Selecting a saved view automatically moves the oneline diagram location to the (x,y) coordinates and zoom level, applies a stored contour, and/or applies hidden layers stored with the selected view.

This dialog can be used for creating a new view, editing an existing view, or deleting an existing view. The **Save View Level Dialog** has the following options:

View Name

This is the name that the view will be stored under in the saved view list. This must be a unique name for each view saved with the oneline diagram. By default this field is blank. To edit an existing view, choose the desired view from the drop down list.

Save Display Information in View

When checked, the view will store an x-y coordinate and zoom level to associate with the view. When the view is selected for display, the diagram will center on the x-y coordinate at the defined zoom level.

X-Coordinate

The x-coordinate for the view. This value will default to the current x-coordinate for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Y-Coordinate

The y-coordinate for the view. This value will default to the current y-coordinate for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Zoom Level

The zoom level for the view. This value will default to the current zoom level for the oneline diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

Save Contour Information in View

If a contour is being displayed when the view is created, you can choose to save the [contour](#) information with the view by checking this box. Saving a contour with a view will display that contour when you switch to that view. You also have the option of saving a blank contour with the view. This is indicated by a Contour Object of 'None' and a Contour Field of 'None.' If you save a blank contour, no contour will be displayed when switching to that view. If you don't save any contour with the view, then any existing contour will be applied when switching to that view.

Contour Object

This section displays the type of object the stored contour pertains to. This field cannot be changed from this location.

Contour Field

Displays the type of value the stored contour pertains to. This field cannot be changed from this location.

Link to Online Display Options Settings

In the [Online Display Options](#) in Simulator, you can save various sets of options with a case for quickly recalling online diagram settings by name. This option allows you to include a custom defined option set by name with the current view. Note that you first must have created and saved a custom set of options for a diagram from the [Online Display Options](#) dialog before this option on the Save Views dialog will be enabled.

Save Hidden Layers

Since Simulator allows the use of [layers](#) to display or hide objects on the diagram, views also can optionally store the layer settings when the view is created. The window below this checkbox will list the layers that are presently hidden on the diagram. When this view is recalled, the layers in this list will be hidden if they were stored with the view.

Save

This button will save a new view or modify the values for an existing view of the name in the **View Name** field.

Delete

This button will delete the currently open view from the saved view list. The dialog information will default to the saved view information of the previous view in the list.

OK

This button will save a new view or modify the values of an existing view of the name in the **View Name** field, and will close the **Save View Level Dialog**.

Save to AXD

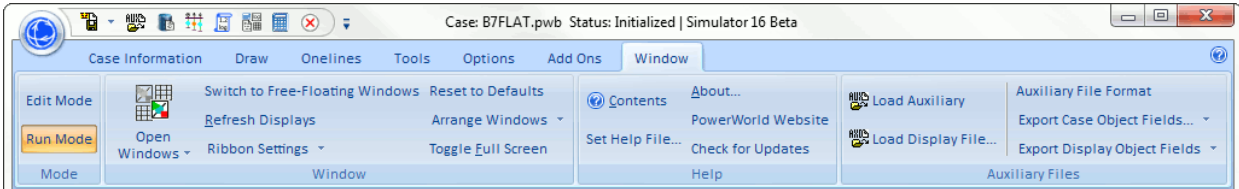
By clicking this button, all the views will be saved to a [display auxiliary file](#).

Load from AXD

This button will load all the views saved in a [display auxiliary file](#).

Window Ribbon Tab

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

Mode Ribbon Group

Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

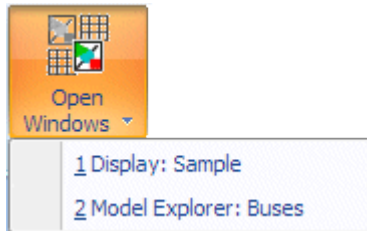
Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

Window Ribbon Group

Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



Switch to Free-Floating Windows (Switch to Standard Container Window)

Choosing this option toggles the program from between two different window environment

1. All diagrams and dialogs are contained within the Simulator program shell
2. All diagrams and dialogs are free-floating on your monitor.

For more information see the [Window Basics](#).

Reset to Defaults

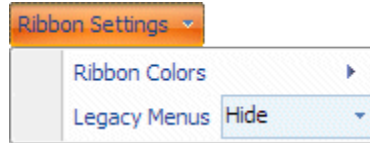
The program keeps track of dialog Free-Floating/Contained Style, Size, and Position. Clicking this button will reset all these to the factory default settings. Nearly all dialogs in will then be reset back to Contained with a default size and position as well. For more information on Style, Size and Position, see [Window Basics](#).

Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

Ribbon Settings Menu

This menu contains options for customizing your ribbon.



Ribbon Colors : You can change the color scheme to either **Blue**, **Silver** or **Black**

Legacy Menus : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

Arrange Windows

The options under Arrange Windows are Tile Horizontally, Tile Vertically, and Cascade. Tile rearranges the open online diagrams such that the total window area is divided equally among all of them and each is completely visible. Cascade Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

Toggle Full Screen

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

Help Ribbon Group

Contents

Opens the [Simulator Options Dialog](#).

Set Help File

Opens the [Case Description Dialog](#).

About

Click this to open the About dialog. The About dialog contains information about the software as well as showing you the Build Date of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About dialog.

There is also an option on the About dialog to Change License Key. This allows selection of a new license key file. Add ons are controlled through the license key. If you are using different versions of Simulator with different add ons, this option provides the mechanism for switching between the different versions of Simulator. You will need a different license key file if you are using different licenses for Simulator that have different add ons.

PowerWorld Website

Click this to open your default web browser to the PowerWorld Corporation Website.

Check for Updates

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

Auxiliary File Ribbon Group

Load Auxiliary

Choose this option to load an [Auxiliary File](#).

Load Display Auxiliary File

Choose this option to load a [Display Auxiliary File](#) to append to the active oneline.

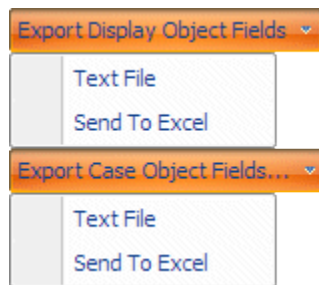
Auxiliary File Format

Opens the [Bus View Oneline](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

Export Display Object Fields Menu

Export Case Object Fields Menu

Choose these options to export a list of fields for each type of object in a case (or a Oneline Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



Online Conditional Display of Objects

Along with supporting zooming and panning, the onelines permit the conditional display of objects. That is, it is possible to specify display objects so they are visible only at particular zooming ranges. This enables the online to show additional details as the user zooms in and fewer details when the user zooms out.

Please note that the zoom levels are defined as percentages. If you want an object to display only between 50% and 150%, you must select 50 and 150 as the zoom level boundaries.

This option is available by assigning objects on the diagram to a [Layer](#). Layers can be defined and set to objects on the [Format Multiple Objects](#) dialog.

Keyboard Short Cut Actions Dialog

The Keyboard Short Cut Actions dialog allows the user to associate a keyboard shortcut to a determined oneline diagram, and even to a particular [view](#) of that diagram. When the keyboard shortcut is pressed, the oneline diagram is opened (if it was closed) and brought to the front. When a view is also associated, then the user is taken to such a [view](#).

To open the Keyboard Short Cut Actions Dialog, select **Keyboard Shortcuts** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Online](#) ribbon tab.

The dialog has the following controls:

Keyboard Shortcuts

Lists all the keyboard shortcuts. When selecting a keyboard shortcut, it will show the *oneline* and the *view* associated to the selected keyboard shortcut. If your keyboard has an extra row of function keys, then you can check the box labeled **Include F13-F24**. If a shortcut has been defined for the respective key, then it will be highlighted and a string will summarize the action of the shortcut

Clear All

Clicking this button will delete all the associations of all the listed shortcuts.

Save Shortcut

Click this button after entering a oneline diagram and eventually a view so they can be associated with the selected keyboard shortcut.

Delete Shortcut

Click this button to delete the associations of the selected keyboard shortcut with any oneline diagram and view.

Create Shortcut

This indicates the keyboard shortcut to which the below actions will be assigned. The available actions are to Open a Oneline and to Open an Auxiliary File

Open Oneline

The following options are available for opening oneline actions:

Oneline and View: Oneline

Shows the path and the name of the oneline diagram which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no oneline diagram is associated to the keyboard shortcut. You can click **Browse** to find the oneline diagram.

Oneline and View: View

Shows the name of the view belonging to the associated oneline diagram to which the user will be taken when the selected keyboard shortcut is pressed. If the field is blank, then no view is associated to the keyboard shortcut.

Action for already open onelines

If the oneline being requested by the shortcut key is already open, then this option determines what should be done. Choices are to close the oneline, bring the oneline to the front, or to open a new oneline.

Where to open oneline

When opening a oneline via a shortcut key, you may do one of the following:

- Separate Window (open the oneline as normal)
- Separate Window – [Toggle Full Screen](#) (open normal and then toggle it to full screen)
- Embed the Active Oneline Window (opens the oneline and embeds it inside the presently active oneline diagram)

Note: if onelines already exist which have been toggled to full screen, and a short cut key is used to open a window that will not be toggled to full screen, then all other onelines present open will be switched to normal mode instead of full screen mode.

Size and Location

This specifies the relative size and location you would like the oneline to be opened at. When opening the oneline normally, the percentage is relative to the size of the container window. When opening the oneline embedded in the active oneline, the percentage is relative to the size of the active oneline.

Make embedded oneline borderless

When [embedding](#) the oneline inside the presently active oneline, this option will make the embedded oneline borderless.

Open Auxiliary File

The following options are available for opening auxiliary file action:

Filename

Shows the path and the name of the auxiliary file which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no auxiliary file is associated to the keyboard shortcut. You can click **Browse** to find the auxiliary file.

Section name

Shows the name of the [script](#) or [data](#) section belonging to the associated auxiliary file which will be executed when the selected keyboard shortcut is pressed. If the field is blank, then no specific section is associated to the keyboard shortcut, so all script and data sections will be loaded.

Create objects if they do not already exist

Check this box to indicate that new objects in the auxiliary file should automatically be created.

Save/Load shortcuts to/from an Auxiliary File

Click the **Save** button to save all the shortcuts to an auxiliary file. Click the **Load** button to load the shortcuts saved in an auxiliary file. The shortcuts saved in the auxiliary file will replace the shortcuts saved in the dialog.

Close

Closes this dialog.

In addition to the shortcuts displayed on this dialog, Simulator also offers standard windows editing shortcuts such as:

Ctrl + x – Cut Command

Ctrl + c – Copy Command

Ctrl + v – Paste Command

Printing Oneline Diagrams

To print a oneline diagram, select **Print Oneline** from the [Application Button](#) or right-click on the oneline background and select **Print Window** from the popup menu. This opens the [Print Options Dialog](#), which you can use to configure the print job, including its size, orientation, border, and title bar. You can even choose to print a oneline to multiple pages using Simulator's multi-page printing.

Print Options Dialog

The Print Options Dialog is used to configure the printing of oneline diagrams (including bus view displays) and of strip charts. To print a oneline diagram, select **Print Oneline** from the [Application Button](#). To print a strip chart, right-click on it to invoke its local menu, and then select **Print Window**. In both cases, Simulator will open the Print Options Dialog.

The Print Options Dialog has three tabs:

Page Layout

This tab sheet contains options for how the diagram will appear on the printed page. It presents the following controls:

Margins

Specify the horizontal and vertical margins in either inches or centimeters.

Scaling

Set this option to **Proportional** to print the oneline diagram or strip chart such that the screen's aspect ratio is maintained. Set this option to **Fit to Page** to force the printout to take up all available space on the page in both horizontal and vertical directions.

Draw Border

Places a border to be drawn around the oneline diagram or strip chart.

The second tab controls the printing of a title bar. Check the **Print Title Bar** box to print a title bar at the bottom of the diagram. Checking this box displays a number of options that allow you to specify various items to include in the title bar. If you don't want to include certain items on the plot, simply leave those fields blank.

The title bar is split horizontally into three sections. If you specify values for all requested items, they will be arranged in the title bar as follows:

Company | Description Line 1 | Date

Department | Description Line 2 | Drawing #

Author | Description Line 3 | Title

Multi-Page

This tab allows the oneline diagram to be "divided" into a specified number of sections for printing the oneline to several pages. A multi-page oneline can be useful for including "close-up" views of areas of a diagram in a report, or the multiple pages can be cut and combined to form a larger printed oneline if a plotter is not available. The **Multi-Page** tab presents the following controls:

Grid Size

Choose this value to subdivide the oneline into an N by N grid. You can then choose which sections of the oneline to print based on the grid overlay. The maximum size is 10 x 10.

Choosing What to Print

A low resolution picture of your oneline is shown on the **Multi-Page** tab. Gridlines are drawn over the diagram to show where the oneline has been subdivided based on the **Grid Size** chosen.

To remove a grid section from the printout, click on the oneline image in the corresponding section of the grid (Please note that the grid size should be more than 1 to select a grid section). The selected section of the grid will become dark to indicate you wish to prevent the section from printing.

To add a grid section back into the print job, click on a darkened section of the grid to allow that section of the oneline to be printed.

The number of pages you have chosen to print will be shown on the dialog. When printing out a multi-page oneline with a title bar, the page number will appear on the title bar in the form "Column Letter"-"Row Number". For example, if the Grid Size is 4, the pages are numbered **A-1** through **D-4**.

Click **Print** to send the document to the printer, or **Cancel** to abort the print. Click **Setup** to view the default Windows printer dialog, which will allow you to specify whether to print the figure in portrait or landscape modes and to set various printer-specific properties.

Printer Setup

Choose **Printer Setup** from the [Application Button](#) to configure the printer using the standard Windows printer setup dialog. This dialog allows you to define which printer to use for printing from Simulator, the size of the printed page, the page's orientation, and additional properties that are specific to the printer you are using.

Contouring

Simulator can create and animate a contour map of various system quantities, such as voltage magnitudes and angles, MW transactions, transmission loading, and real and reactive load. Such displays resemble a contour map of temperatures like you might see shown on a weather forecast. Contouring can significantly improve understanding of a large interconnected system, helping identify congestion pockets and Mvar-deficient regions and providing an overview of how power flows through the bulk power system.

The [Contour Options Dialog](#) controls Simulator's contouring capabilities. To access it, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.

Contouring is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically removes the contour from the oneline diagrams. You can still choose to reapply the contour while in edit mode however.

Contouring Options

The Contour Options Dialog allows you to draw contour maps of many system quantities, such as bus voltages or angles, transmission line and interface MVA loadings, and transmission line and interface [PTDFs](#).

To access this dialog, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring > Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. The Contour Options Dialog has three tabs: the [Contour Type](#) Tab, the [Contour Type Options](#) Tab, and the [Custom Color Map](#) Tab.

Contour Type

Contour Type options can be set from the Contour Type tab on the [Contour Options dialog](#).

Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). See NOTE at the end of this help topic.

Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

Data Points per Line

This specifies the number of data points used to represent the graphical line. The contouring algorithm is based on values at specific coordinates in the oneline space. When contouring transmission line or interface objects, the line is represented by a series of points spaced equally along the graphical line.

No Data Color

This setting allows you to choose the contour color used for parts of the contour for which there are not objects nor data for the type of contour selected. The choices for No Data Color are Specific Color, Color Map Percentage, Background Color, and Transparent. By default, Specific Color is selected and set to white. If you wish to change the specific color to use, click on the color box to the right and choose a different color from the popup dialog. If you select Color Map Percentage, the Color Map % field will become enabled, and you can select a value from 0 to 100. The value you enter will associate the No Data Color with the color located at that percentage in the selected color map. If Background Color is chosen, the No Data Color will always be whatever color has been set as the normal background color for the oneline diagram. Lastly, if Transparent is chosen, the parts of the contour for which there are not objects nor data for the type of contour selected will be transparent and any objects on the oneline diagram will be displayed as usual.

Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

Title

Title for the color key.

Entry Labels

Units of the contoured value displayed on the color key.

Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

Scalar

Multiplication factor that can be applied to the values when drawing the color key. Normally this value should be 1.0.

Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent. This option only applies to discrete color maps.

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

Reverse Color Map Colors

Check this check-box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Use absolute value

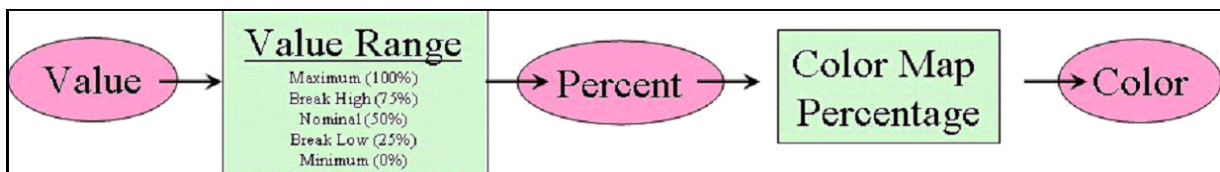
Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

Ignore Above Max

Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

Values

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.



User may define their own color maps which convert percentage to color, but they may also define color maps which directly translate values into colors without the intermediate calculation of a percent. For more information on this see the creation of a [Custom Color Map](#).

The value range values are described as follows:

Maximum The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

Ignore Below Min

Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

Ignore Zero Values

Check this box to completely ignore zero values in the contour. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so. If there is a need to ignore values that are almost but not exactly zero, then create an Advanced Filter to Filter Object Values.

Interpretation

This combo box specifies how to interpret the values of the data points. The options are:

Fixed Values The data point values are not modified. The maximum, minimum, nominal, and break values are the ones entered directly in the units of the value being contoured. This is what should normally be used.

Dynamic Values The data point values are not modified. However the maximum, minimum, nominal, and break values are determined dynamically from the data point values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High = (Max + Average)/2; and Break Low = (Min + Average)/2.

Standard Deviations All the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on.

Percentiles All the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

Save Contour Image with Oneline

Checking this box will allow a displayed contour to be saved with a oneline diagram. If a contour is saved with a oneline diagram, the next time the oneline diagram is opened the contour will automatically be redrawn as well. The displayed contour is saved as a Bitmap image.

Continuously Update Contours

Normally contouring is only done on a snap shot of the power system state. However, you can also set Simulator to automatically update the contour every time the display is redrawn. In this way, an animation of the contour can be created. If you would like to create this animation, simply check the Continuously Update Contours checkbox. Note, however, that this will slow down the animation of the display, as the program must recalculate the contour at each step. If this slows down your display too much, try lowering the contour resolution to speed it up.

Note Regarding Values

Contours of most values create an image where the color around a data object is primarily related to the value of only that object. Some values however create "density-like" contours, where the color is related to the sum of the data object's values nearby. These include:

- Bus / Load MW
- Bus / Load Mvar
- Bus / Load MVA
- Bus / Cust Expr (Density)
- Area / Pos Spin Reserve
- Area / Neg Spin Reserve

Contour Type Options

Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

If multi-section lines objects exist on the oneline diagram and the type of object to contour is Line, the contour for the multi-section line object will reflect the series of AC lines that it represents. If the type of object to contour is Bus, the locations of intermediate buses will be contoured.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). Note: see [Contour Type](#) for more information.

Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

Influence Region

This track bar determines how far away each data point influences the contour image. A larger influence region results in each data point influencing more of the contour at the expense of longer screen refresh times.

Use Dynamic Influence Region

Dynamic influence distance determines how far out the contour should go when determining which buses influence the contour value for a screen point. The actual distance is the minimum of either 1) a common value for all screen points that depends upon user parameters [e.g., the dynamic region points value]), and now 2) the distance that includes the number of buses associated with the dynamic influence field. The primary reason for this option is speed, particularly when zoomed in on dense portions of the display. You should not see much impact on the contour itself.

Kind of Value

This option allows you to choose to contour based on the Actual Value or the Density Value. The Actual Value uses the weighted average of the data for computing the contour. The Actual Value method is most commonly used for contouring in Simulator. On occasion, the Density of Values method does not work as well. One example is contouring generator MW values. If you have four buses in close proximity, each with 100 MW of generation, and compare the contour with a single bus with 400 MW of generation, the contour based on the weighted average will look drastically different, despite the amount of generation being the same in each region. Using the Density Value option will correct the disparity, and the contour around these two different groups of generation would look basically the same.

Use Fade to Value

Checking this check box will allow to use the Fade to Value and Begin Fade Percentage options.

Fade to Value

The value to which a data point's value fades as it moves away from its location.

Begin Fade Percentage

While moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

Contour Resolution

This value determines the relative resolution of the contour. Increasing the contour resolution increases the level of detail represented on the map but will lengthen screen refresh times. Reducing the screen refresh time will yield less detail and shorter screen refresh times.

Low Resolution Display when Zooming or Panning

Do Low Resolution Speedup

Check this option to enable drawing contours in low resolution while zooming or panning. Low resolution images are drawn much faster than high resolution images.

Low Resolution Normalized Percent

Set a percentage of resolution between 0.01 (1%) and 1 (100%). A smaller value translates to a lower resolution image.

Time Delay in Seconds before Redrawing at Full Resolution

Set the number of seconds Simulator will pause before returning the contour to full resolution following zooming and panning.

Minimum Resolution in Pixels for Low Resolution Displays

Specify the minimum resolution allowed (in pixels) when Simulator draws the low resolution image.

Use GPU to accelerate contour calculation

As it describes, checking this option will use the GPU processor to perform contour calculations that will accelerate computation time.

Use 32 bit texture format

As it describes, checking this option will use a 32 bit texture format for contours.

When Animating, Only Update Contour if Something Changed

As it describes, checking this option will suppress redrawing the contour during animation unless one or more of the contoured values changes.

Custom Color Map

Custom Color Maps can be defined from the Custom Color Map tab found on the [Contour Options dialog](#). Custom color maps allow you to define completely new color maps for create new color maps from existing default color maps.

Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

Reverse Color Map Colors

This check-box reverse the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

Brightness

Modify the brightness track bar to change the brightness of the color map.

Make Discrete Color Map

Check this check-box to make a discrete color map, that is without having smooth transitions between colors.

Contour Type Values to Use

These check-boxes signify which values from the Contour Type Values Tab are used by the Color Map. There must be at least two contour type values checked.

Color Grid

The color grid on the right side of this page allows you to change the colors for each percentage breakpoint. In addition, you can add or delete breakpoints as well.

To change the color for a specific breakpoint, simply left-click on the color for the breakpoint you want to change. The Color dialog will open, and you can choose a new color for that breakpoint. Setting the Transparent field to YES will make the contour transparent for data values that meet the specified percentage breakpoint. Setting the Transparent field to NO will show the data values with the specified color. To change a value in the Transparent field, left-click on the value and it will toggle between YES and NO.

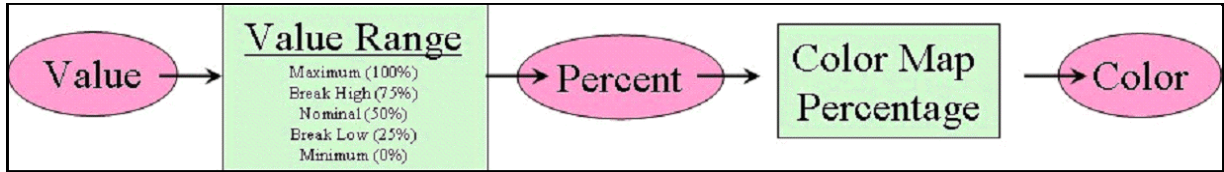
To add a breakpoint, right-click on a breakpoint position above or below where you would like the new breakpoint inserted. A popup menu will open, and you can select Add Above or Add Below, depending on where you wish the new breakpoint to be. Simulator will insert the breakpoint, and will automatically set the color and percentage at the midpoint between the two breakpoints above and below the inserted breakpoint. You can then click on the color to change it, or click on the percentage to type in a new value. The new percentage value should be between the values of the adjacent breakpoints.

To delete a breakpoint, right-click on the breakpoint you wish to delete, and choose Delete from the popup menu.

Color By...

This option allows the user to interpret the breakpoints values as percentage values or as direct values.

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.



Color maps which directly translate values into colors without the intermediate calculation of a percent may also be created. This is done by setting Color By to *Value*.

Save As...

To save the present color map as a new color map, click this button. Then specify a name for the new color map.

Save

To save changes that have been made to the present color map, click this button.

Rename

To rename the present color map, click this button.

Delete

To delete the present color map, click this button.

Store Color Maps in File

To store all custom color maps in a file for loading into another case, click this button. If you have saved any custom color maps with the current case, you will be prompted to choose a file name and location for saving the custom color maps. Color Maps are saved in the [Auxiliary File Format](#).

Load Color Maps from File

If you have created custom color maps in a different case and saved them to a file, you can click this button to load those color maps into your current case. Color Maps are loaded in the [Auxiliary File Format](#).

Functional Description of Contour Options

Functional Description of the Contour Options

The previous help topics discussed basic contour options in the order they are arranged on the dialog. The dialog is arranged so that the most important options are on the first tab and other on the Contour Type Options tab. This help topic discusses the options in a manner which better describes how the contour is actually created. A contour is calculated generally by a five-step process

1. Build a list of all possible data points (graphical locations)
2. Remove items that meet criteria from the list of data points
3. Assign a value to each data point
4. Calculate "virtual values" on a grid of points
5. Convert each "virtual values" into a color to create the contour image

Step 1: Build a list of data points (graphical locations)

The following options determine a list of potential data points (could also be called graphical locations) which will be used to calculate the contour image.

Object - Simulator can contour several different values. To specify what Simulator should contour, first choose the type of object. This corresponds to the type of display object that is drawn on the oneline diagram. For instance if you want to contour a substation value, then you must have substations drawn on your diagram. This selection narrows the set of quantities that can be contoured, which is specified in the Value dropdown box.

Data points Per Line - The contouring algorithm for lines is no different than for points, except that each line is represented by several points. For objects which are represented by graphical lines, this option will specify the number of data point which should be used to represent the line.

Step 2: Remove items that meet criteria from the list of data points

After a complete list of potential data points is made, there are then several options for filtering out things from this list which you do not want to effect the calculation of the contour image.

Filter - If you wish for the contour to be limited to only certain devices that meet specific criteria, click on this button to define an Advanced Filter for the contour

Ignore Above Max - Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image

Ignore Above Min - Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image

Ignore Zero Values - Check this check-box to completely ignore values that are zero.

Step 3: Assign a value to each data point

After Step 2, a list of potential data points has been created and a value must now be assigned to each data point. The following options specify how this is done.

Value - select the quantity to contour from the Value dropdown box or click the Find Value button to find the desired field.

Use absolute value - This check-box will modify the Value specified so that it uses the absolute value.

Interpretation – When interpretation is set to either Standard Deviations or Percentiles, then the value of the data point will be modified.

If **Standard Deviations** is chosen, then all the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on. Similar,

If **Percentiles** is chosen, then all the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

Step 4. Calculate "virtual values" on a grid of points

After Step 3, we now have a list of data points and a value assigned to each data point. We now must create a grid of points that represents "virtual values" throughout the entire graphical space. The values on this grid will be calculated based on the data point values

Contour Resolution - setting determine the size of the grid of points which will be superimposed on the present oneline diagram. The higher the resolution the more number of grid points will be used to represent the contour (and thus make the calculation of the contour image slower)

Influence Region - each data point value will effect only the grid locations that are "near" it. The distance that is considered "near" is determined by the Influence Region. Setting the influence

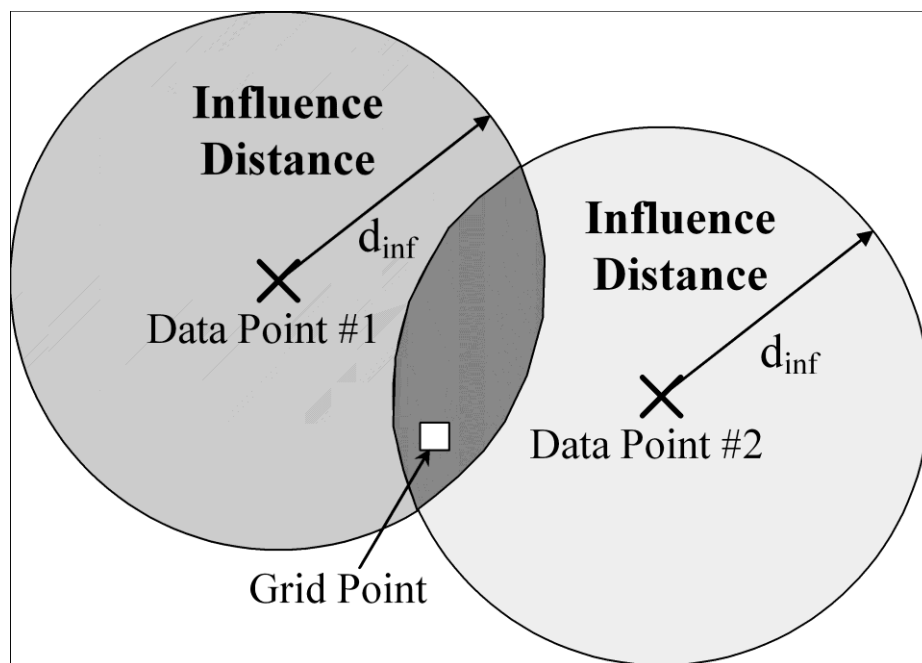
region higher will result in each data point effecting a larger portion of the contour image (and thus making the calculation of the contour image slower)

Kind of Value - The calculation of the virtual value at a particular grid point is done by first building a list of data points that are within the "influence distance" of the grid point. The Kind of Value setting determines what process is used to calculate the virtual value from this list of values.

Actual Value (Weighted Average) - The virtual value is calculated as the weighted average value, weighted by the distance from the grid point. This means that virtual value half-way between two data points will be the average of the two data point values.

Density of Values (Weighted Sum) - The virtual value is calculated as the sum of value that within the influence region. This means that virtual value half-way between two data points will be the sum of the two data point values.

Actual Value (Only Closest) - The virtual value is assigned as the value of the closest data point.



Fade to Value and Begin Fade Percentage - In the middle of a contour image the colors look consistent because the virtual values are calculated using a good number of data points which surround it. Sometimes at the edge of a contour image however, the colors can become skewed because there are very few data points influencing it. Using a fade to value can help this situation, but will skew the entire contour image in general instead. Without using the fade to value, the raw data point values will be used when calculating virtual value.

When using the fade to value, while moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

Step 5. Convert each "virtual values" into a color to create the contour image

After Step 4, a grid of virtual values has been calculated. At this point, must specify how these numbers map to colors. This done through the user of a Color Map.

Color Map - Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image. A user may also define additional color maps by going to the Custom Color Map Tab.

Brightness - this value is used to brighten or darken the colors specified in the color map.

Values - These values along with the color map define how to convert your values into a color for the contour. The values are:

Maximum - The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

Break High - This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

Nominal - This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

Break Low - This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

Minimum - The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Interpretation - Most frequently this option will be set to **Fixed Values** meaning values for maximum, break high, etc... are entered directly in the units of the value being contoured. If Interpretation is set to **Standard Deviations** or **Percentiles** then as mentioned earlier the values mean something different. The options **Dynamic Values** will process this list of data point values and automatically set the values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High = $(\text{Max} + \text{Average})/2$; and Break Low = $(\text{Min} + \text{Average})/2$.

No Data Color - It is likely that some the grid points will not be within the influence of any of the data points. The color of these points is determined by no data color setting.

Dynamic Formatting Overview

The [Dynamic Formatting dialog](#) will allow the specification of how a graphical object will be rendered depending on the power system object that is representing. The graphical object refers to objects in the oneline diagram, in bus and substation views, as well as to some parameters of the case information displays.


The case and the one-line diagrams will have each a list of Dynamic Formatting settings. The Dynamic Formatting settings defined in the case are always applied to bus and substation views, and optionally they can be applied to the case information displays. The one line diagrams will use its own settings, but optionally can use the general settings defined with the case.

In general, the oneline dynamic formatting settings have a higher priority, followed by the case dynamic formatting settings, and this can't be reversed. Also, inside each of the list of dynamic formatting settings, a priority can be specified, so that objects can be rendered according to the settings with the highest priority.


In the case of the oneline diagrams, the dynamic formatting settings will only be applied during Run Mode. However, these settings can be modified at any time, without regard for the mode.

Dynamic Formatting Dialog

The Dynamic Formatting dialog for the active oneline can be accessed by:

- Selecting **Dynamic Formatting** >  **Active Oneline** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (Active Oneline)** from the [oneline local menu](#), or

The Dynamic Formatting dialog for the general case information displays, bus and substation views, and for all onelines can be accessed by:

- Selecting **Dynamic Formatting** >  **Case Info / All Views And Onelines** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (All Views)** from the [bus view local menu](#) or the [substation view local menu](#), or

This dialog presents the following options:

Set Format Priority

It is possible to have multiple dynamic format definitions created and stored with a case and/or active oneline diagram. This option allows you to set a format priority order for all of the dynamic format definitions, so that any diagram objects that meet more than one dynamic filter will use the settings of the dynamic filter with the highest set priority.

Allow Oneline to use dynamic formatting defined with case

This option will only be available when the dynamic formatting settings correspond to an active oneline. It indicates whether or not the oneline diagram will use the dynamic formatting definitions specified in the case.

Object Type

The type of power system object with graphical representation, such as a bus, a load, a generator, etc.

Filter Criteria

The filter that applies to the object type defined. The Dynamic Formatting settings will apply to the graphical object only if the corresponding power system object meets the specified filter. If the criteria box is empty, the Dynamic Formatting settings will assume that all the objects of the specified type meet the criteria.

Formatting Active

If this option is unchecked, the dynamic formatting definition will be ignored when the objects are rendered. Otherwise, it will be applied if there are graphical objects whose characteristics match the rest of the characteristics defined in this dialog.

Force visibility

When this option is checked, the objects will be displayed (assuming the dynamic formatting applies to them) independently of the visibility of the layer to which such objects belong, and with no regards of the low and high zoom levels.

Show Lookup Tables

Checking this box will display the **Use Lookup** options with the characteristic settings. The **Use Lookup** option with certain field characteristics allow you to define a table of different field values and corresponding characteristic values. If you check Show Lookup tables, and then check the Use Lookup field next to one of the characteristics, you will open the Lookup definition table. This table allows you to select a field for the Object Type you have selected. This field is the field that you define values for in the left column of the lookup table. To insert entries in the lookup table, right-click in the table grid and choose Insert from the popup menu. This will place a default record in the table. Modify the Field Value that Simulator should look for in the table, and a corresponding Characteristic Value that should be used for the field value given. For example, you could use a lookup table with Line Thickness for transmission line objects, where the thickness of the line would vary based on the amount of flow on the line. Different thicknesses could be assigned to different levels of flow using a lookup table.

Context

In the Context Objects, the user will specify what specific type of graphical objects the dynamic formatting will apply to. This list view will be populated with the Case Information Display object (if the dynamic formatting definitions set correspond to the general case), plus the several graphical objects related to the Object Type. (For buses, for example, it will include the graphical bus, the bus fields, and the bus gauge).

Fields

In the Fields list view, the user will be able to select which fields the dynamic formatting settings will apply to. This view will be populated only for those context objects with fields, such as the Case Information Display and the Object Fields. This list is only visible when you choose certain types of context objects in the context objects list.

Show Only Commonly Used Fields

If this option is checked, only a reduced list of selected fields will be displayed. Otherwise, all the fields belonging to the object will be shown.

Characteristics

The characteristics that the user will be allowed to modify dynamically. These include line thickness, style, color, and background color; font name, size, color, and style; highlight color; surround shape, color and thickness; color and magnification of the 'X' on top of the objects; and blinking color and interval.

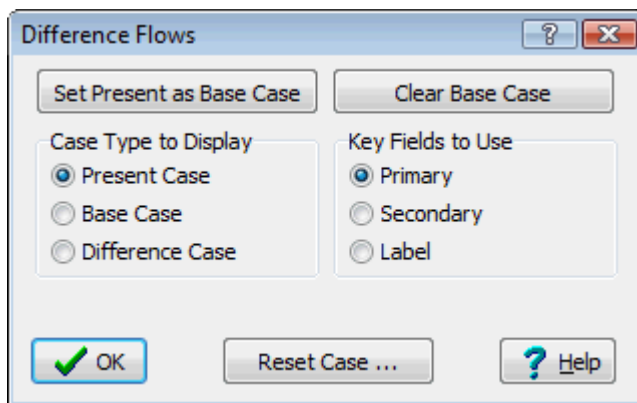
Difference Flows

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, Difference Flows can be used to show the difference in transmission line flows and bus voltages resulting from a contingency or a change in power transfer between two areas. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

For more information about how to use Difference Flows see [Using Difference Flows](#).

The Difference Flows Dialog can be accessed by:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows > Difference Flows** from the [Other Tools Ribbon Group](#).
- Selecting **Difference Flows** from the [online local menu](#).
- All Difference Flows Actions are controlled via the Difference Flows Dialog or the drop-down list on the Difference Flows toolbar button.



Case Type to Display

Use of this feature affects all aspects of the Simulator environment. When using the Difference Flows tool, information shown on online diagrams, case information displays, and [power flow lists](#) is governed by which of the three [Difference Flow Case Types](#) (base, present or difference) is currently being displayed.

Key Field to Use

The difference flows feature must make a comparison between the Present Case and the Base Case to determine which objects between these two case match one another. The Key Fields to Use specifies whether this matching is done using Primary Keys (bus numbers), Secondary Keys (bus names/Nom kV pair), or by Label.

Difference Flows: Case Types

When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three case types is currently being displayed. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

The case types available are:

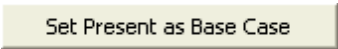
- **Base Case** – A solved power system that serves as the reference for the difference flows tool. To establish a base case, set up a solved power system corresponding to the desired operating point. Open the Difference Flows Dialog and click the button labeled **Set Present as Base Case**.
- **Present Case** – The operating point used in the Difference Flows comparison. **Note:** The Present Case must have the same numbering scheme as the Base Case for proper operation of the Difference Flows tool. See [Topological Differences](#) for more information.
- **Difference Case** – The difference between the Present Case and the Base Case values. The values displayed in the Difference Case are established using the Base Case as the reference. When showing status fields for branches, generators, etc... fields will show up as either OPEN/CLOSED or CLOSED/OPEN if the status changed. The status in the present case will be listed first. A similar format will be used for fields such as generator AGC Status and other text fields.

To toggle between the different case views, open the [Difference Flows Dialog](#) and select the desired case type. Alternatively, you can click the **Difference Flows** button in the [Other Tools Ribbon Group](#) on the [Tools](#) ribbon tab and select the desired case type.

Using Difference Flows

To use the Difference Flows tool:

- Set up a solved power system corresponding to a desired operating point. This operating point will be defined as the [Base Case](#).
- Select **Difference Flows** from the Run Mode ribbon group on the [Tools](#) ribbon tab, or select **Difference Flows** from the [online local menu](#) in Run Mode.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the Base Case.



- Define the operating point ([Present Case](#)) for which to perform the difference flows comparison. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case selecting **Open Case** from the [Application Button](#). In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- See [Difference Flow Case Types](#) for information on toggling between case views. The currently displayed case type is shown in the [PowerWorld Simulator Status Bar](#). When viewing either the Base Case or Difference Case, the corresponding status bar display will be highlighted.

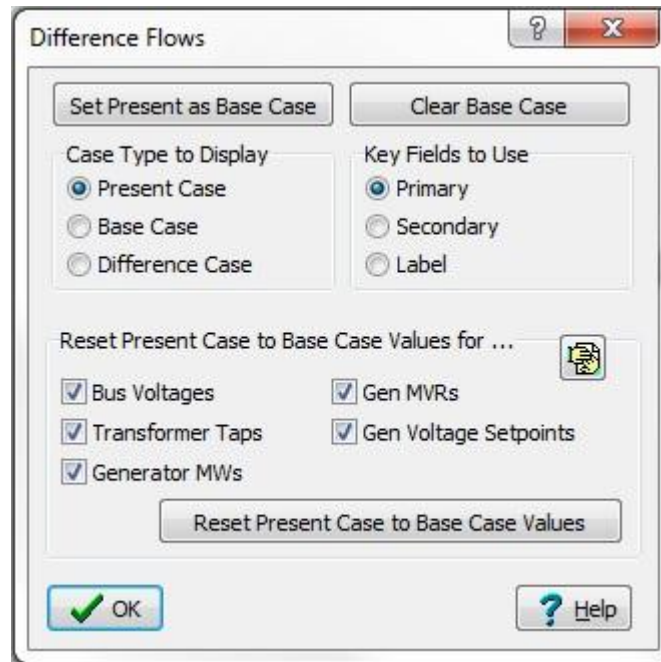


Note that the Difference Flows tool can only be used in Run Mode and the status bar will not display a case type while in Edit Mode.

When showing the Difference Case, most of the fields shown on the onelines and case information displays show the difference between the present value and its Base Case value. For example, on the Generator Records Display, an entry of 0.0 in the MW field indicates that the real power output of the generator did not change. An entry of 10.0 in the MW field indicates that the present real power output of the generator is 10 MW greater than it was in the Base Case.

At any time during a simulation, you can set the present case as the Base Case by clicking the corresponding button on the Difference Flows Dialog.

Conversely, if you have made changes to the present case, and you wish to revert to some or all of the base case values, you can click on the **Reset Case...** button. When you click this button, the dialog will expand to show you options for resetting specific types of values to their base case values.



Expanded Difference Flows Dialog

Once you have indicated which types of values you want to reset (by default, all are selected), you can click on the **Reset Present Case to Base Case Values** button to complete the process of reverting to base case values. Note: once you have finished resetting to the base case values, you can hide the Reset options by clicking the Hide Reset Button in the upper right hand corner of the Reset options panel.

The oneline diagrams and case information displays cannot indicate structural differences in the case very well, such as the addition or removal of a device. To identify such differences, make use of the [Present Topological Differences from Base Case](#) tool to identify topology differences.

Present Topological Differences from Base Case

The Present Topological Differences from Base Case option provides the users a way to compare the topological differences between two difference cases in Simulator.

To compare two cases topologically, you must load the first case you wish to use as the reference into Simulator. Use the [Difference Flows](#) tool to set the case as the base case in memory. Once the reference case has been stored as the base case, open the second or comparison case into Simulator. Now if you check the [Tools](#) ribbon tab, and choose the **Difference Flows Menu** on the [Other Tools Ribbon Group](#), you will see the option **Present Topological Differences from Base Case**. Choosing this option will open the dialog to display the topological differences.

Present Case Topological Differences from the Base Case

Summary | Elements Added | Elements Removed | Elements In Both | Create Bus Swap List

Below is a summary of the comparison between the present case and the base case saved from the Difference Flows Dialog.

Element Type	New	Removed	Both
Bus	0	0	9
Load	0	0	3
Switched Shunt	0	0	1
Generator	0	0	3
Branch	0	0	9
DC Line	0	0	0
Area	0	0	1
Zone	0	0	1
SuperArea	0	0	0
Transformer	0	0	3

☒ Assume base case Areas/Zones which are not in present case meet the Area/Zone Filters

Save and Send Option

All Lists

☐ Use Area/Zone Filters when saving to Auxiliary File

Send To Excel | Save to Text File | Save To EPC File | Save To Aux File | Load Aux File

Close

Topological Differences Dialog

The Topological Differences Dialog contains five tabs. All but the Summary tab are instances of Case Information Displays and thus exhibit characteristics and controls similar to other displays.

Summary

A listing of the types of objects in the case. The first column displays the number of NEW devices in the comparison case that were not in the base case. Alternately, the Removed column displays the number of devices in the base case that do NOT exist in the comparison case. The third column simply displays the number of items that were matched between the two cases.

Elements Added

Tabular listings of all objects that exist in the comparison case, but not in the reference case.

Elements Removed

Tabular listings of all objects that exist in the reference case, but are not present in the comparison case. A right-click option from the pop-up menu on the Elements Removed tables allows for saving the removed elements selected to a GE EPC file format.

Elements in Both

Tabular listings of all objects present in both cases.

Create Bus Swap List

Used for setting up a bus renumbering list. It is possible that discrepancies in topology between the comparison case and the reference case can be due to a difference in bus numbering between the two cases. [Renumbering the buses](#) in the comparison case may take care of most of the topology discrepancies that are reported in this instance.

Data Objects and Fields

The Save and Send Options on the topological difference dialog makes it possible to save Elements Added, Elements Removed, and Elements In Both to file. In addition to these options, special objects and an object field are available to make accessing these objects easier using script actions and will allow the user to customize the fields stored in the file. When the removed elements are saved in the GE EPC file format the elements are saved with a status value of -4.

To provide easier access to the objects that have been identified as being removed when viewing the topological differences, special object types have been created that allow these objects to be accessed via script commands. These objects provide access to the Elements Removed. All of the object names start with *Removed* with the rest of the name just being the usual object name. To identify buses use the object name *RemovedBus*. To get generators use *RemovedGen*, etc. These objects can be accessed through the [SaveData](#) script command and [GetParametersMultipleElement](#) through SimAuto. These object types can only be stored to an auxiliary file. They cannot be read back in, but, the file headers and object types can be modified to make use of the data or the data can simply be used as documentation of case differences.

A field is available for all object types that can be used with the difference flows tool. This field is found under **Difference Flow\In Diff. Flows Base Case?** and indicated whether or not an object was in the difference flows base case. This field is useful for filtering only the new elements that are only in the present case, field value = NO, or all of the elements in both the present and base case, field value = YES. This will make it possible to use script actions or filters to return only the Elements Added or Elements In Both.

Limit Monitoring Settings

Simulator offers many tools to study the capabilities of a power system. Examples include:

- [Contingency Analysis](#)
- [Available Transfer Capability \(ATC\)](#)
- [Optimal Power Flow \(OPF\)](#)
- [Security Constrained OPF \(SCOPF\) Overview](#)
- [PV Curve and QV Curve Tool](#)

All of these tools make extensive use of power system limits. Limits for various power system elements include:

- MVA (or Amp) limits on transmission lines and transformers
- MW limits on Interfaces
- High and low voltage limits for Buses

The accuracy of all these limits is very important, as is the specification of which limits should be monitored. While ensuring the accuracy of input data such as power system limits must be left to the user, PowerWorld Simulator provides several ways to specify which limits should be monitored. Limit Monitoring is specified according to settings for the Area, Zone and Limit Group to which the power system element belongs. A power system element is monitored only if ALL of the following conditions are met.

Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The [Limit Monitoring Settings Dialog](#) gives you ability to specify all these settings to setup the limits you want to monitor, and thus enforce in the various tools that Simulator provides.

Limit Monitoring Settings and Limit Violations Dialog

The Limit Monitoring Settings Dialog gives you ability to specify which limits you want to monitor, and thus enforce in the various tools that Simulator provides. In general, keep in mind that a bus, transmission line/transformer, or interface's limit is monitored only if the following conditions are met.

Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The following fields are shown on the display:

Elements To Show

Change this value to modify which elements are displayed in the Buses, Lines, and Interfaces. Set it to...

All Elements to show all Buses, Lines and Interfaces regardless of monitoring settings.

Monitored Elements to only display Buses, Lines and Interfaces that meet the conditions for monitoring. See [Limit Monitoring Settings](#) for more information on setting which values are monitored.

Violating Elements to only display Buses, Lines and Interfaces that are violating. See [Limit Monitoring Settings](#) for more information on setting what is considered a violation.

Number of Violations

The following fields provide summary information on the types of violations that are currently present in the case:

High Voltage Buses and Low Voltage Buses

Shows the total number of bus voltage magnitude violations. These violations are shown on the **Bus Voltage Magnitude Limit Violations Display**.

Low Voltage Suspects

Shows the total number of buses whose voltages have fallen below a designated internal threshold to indicate a low voltage solution is being reached in some location in the system.

Line/Transformer Violations

Shows the total number of line/transformer violations. These violations are shown on **Line and Transformer Limit Display**.

Interface Violations

Indicates the total number of interface violations. These violations are shown on the **Interface Violations Display**.

Limit Group Values

This part of the dialog shows information about the Limit Groups with greater description shown below.

Save/Load Monitoring Settings

The Save and Load Monitoring Settings buttons allow the user to save the current set or load a previously saved set of Limit Monitoring Settings options. The options are saved in a [PowerWorld Auxiliary File](#) text format.

Do not monitor radial lines and buses

Check this box to ignore limits on radial lines and buses throughout the case. In this instance a radial bus is defined as a bus that is connected to the rest of the power system by a single in-service transmission line. A radial line is defined as a line connected to a radial bus. This means that in a 5 lines in series that connect to a radial load, only the last bus and last line will be considered radial.

If you would like to better determine which lines are "radial" in a more traditional sense, use the feature [Branches that Create Islands](#) that allows you to easily generate a list of branches that if taken out of service will split an existing island into two. These represent lines which may be considered radial.

Tabs on the Limit Monitoring Dialog

The tabs on the Limit Monitoring Settings Dialog allow you to change these settings. These tabs contain tables showing lists of the respective elements. Since these tables are another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the local menu, you can print the violations, [copy the violation records](#) to the Windows clipboard for use with another application, modify the format and content of the violations listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

Buses Tab

Each tab shows a list of the respective type of power system element. The important columns include:

Number, Name: Bus number between 1 and 99,999, and its alphanumeric identifier.

Area Name: Alphanumeric identifier of the bus' area.

Monitor: toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Limit Group: toggle this value to specify which Limit Group the element belongs to

PU Volt: Bus' per unit voltage magnitude.

Volt (kV): Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

Limit Low PU Volt: Bus' individual low voltage limit used when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the low per unit voltage limit specified in one of the eight low voltage limit sets that can be defined with the bus. Which voltage limit set's value is displayed depends on which limit set is specified as the "Bus Low Rate Set" in the limit group to which the bus is assigned.

Limit High PU Volt: Bus' individual high voltage limit used when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the high per unit voltage limit specified in one of the eight high voltage limit sets that can be defined with the bus. Which voltage limit set's value is displayed depends on which limit set is specified as the "Bus High Rate Set" in the limit group to which the bus is assigned.

Contingency Limit Low PU Volt: Bus' individual low voltage limit used during a contingency when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the contingency low per unit voltage limit specified in one of the eight contingency low voltage limit sets that can be defined with the bus. Which contingency voltage limit set's value is displayed depends on which limit set is specified as the "Bus CTG Low Rate Set" in the limit group to which the bus is assigned.

Contingency Limit High PU Volt: Bus' individual high voltage limit used during a contingency when the bus is set to use specific voltage limits other than the default voltage limit defined in the bus' limit group. This field is not enabled unless the "Use Specific Limits" option for the bus is set to YES. The value will then reflect the contingency high per unit voltage limit specified in one of the eight contingency high voltage limit sets that can be defined with the bus. Which contingency voltage limit set's value is displayed depends on which limit set is specified as the "Bus CTG High Rate Set" in the limit group to which the bus is assigned.

Note: If **Elements to Show** is set to *Monitored Elements*, then only elements which meet the conditions for monitoring will be displayed in these lists.

If using the [Integrated Topology Processing add-on](#), an **Only show the primary bus for each superbus** checkbox will be available. Checking this box will limit the buses shown to only those that are primary buses. This provides a quick means of removing the clutter of redundant data. The primary buses and superbuses are determined from the base case topology.

Lines Tab

Each tab shows a list of the respective type of power system element. The important columns include

From Bus Number and Name: Bus number at name at the from terminal of the line. For transformers, the from bus is the tapped side. Right-clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

To Bus Number and Name: Bus number at name at the to terminal of the line. For transformers, the to bus is the untapped side. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

Circuit: Two-character identifier used to distinguish between multiple lines joining the same two buses.

Monitor: toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Limit Group: toggle this value to specify which Limit Groups the element belongs to

Used Limiting Flow, Limit, Used % of Limit: The flow at the end of the branch selected for measurement, and its MVA or Amp limit. The percentage equivalent of the flow to its limit is given in the Used % of Limit column.

MVA or Amps?: Units used with the Max Flow and the Limit field. All flows are expressed either in MVA or amps.

Note: If **Elements to Show** is set to *Monitored Elements*, then only elements which meet the conditions for monitoring will be displayed in these lists.

Interfaces and Nomograms Tab

Similar to the Buses and Lines Tabs shown above. The important columns are the **Monitor** and **Limit Group** columns

Area Reporting and Zone Reporting Tabs

These tabs display all the Areas and Zones in the system. The important columns are

Report Limits: toggle this between **YES** and **NO** to set whether the specific element should be monitored.

Report Min kV and Report Max kV: Only buses and lines within this kV range will be monitored.

Modify/Create Limit Group Tab

Every power system elements belongs to a single Limit Group. By default, all power system elements are in the same Limit Group that is named "**Default**". New limit groups can be added by right-clicking in the table and choosing **Insert** from the popup menu. Additional limit groups give the flexibility of assigning devices to different groups, where each group can have its own set of defined limit information.

The limit group stores important values regarding the enforcement and monitoring of the power system elements within the Limit Group. The values are:

Disabled: Set the value to **YES** to ignore all power system element limits in the Limit Group. Set the value to **NO** to monitor limits according to the settings of the Limit Group.

Branch Percentage: The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

Line Rate Set: You may define eight different ratings to transmission lines or transformers (for more information see [Line/Transformer Information](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

Contingency Line Rate Set: This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

Amps or MVA: Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$AmpRating = \frac{MVARating}{\sqrt{3} * BaseKV} * 1000$$

Also note that the reported Amp% and MVA% will be different by a factor of the per unit voltage at the bus terminal at which the limiting flow is determined. This is often a point of confusion as the expectation might be that the percent loading would be the same regardless of which reporting method is used:

$$Amp \% = \frac{MVA \%}{V_{pu}}$$

When doing contingency analysis and the Power Flow Solution Options are set to [Use the DC Approximation in Power Flow](#), this option will be ignored. Line limits will always be reported in MVA when using the DC approximation.

Interface Percentage: The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

Interface Rate Set: You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

Interface Contingency Rate Set: This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

Low PU Volt: Buses will be flagged as violated if they fall below this per unit voltage.

High PU Volt: Buses will be flagged as violated if they go above this per unit voltage.

Contingency Low PU Volt: Buses will be flagged as violated during contingencies if they fall below this per unit voltage.

Contingency High PU Volt: Buses will be flagged as violated during contingencies if they go above this per unit voltage.

Bus Low Rate Set: This field value indicates which bus voltage rating set should be used for normal (non-contingency) low voltage violation reporting.

Bus High Rate Set: This field value indicates which bus voltage rating set should be used for normal (non-contingency) high voltage violation reporting.

Bus Contingency Low Rate Set: This field value indicates which bus voltage rating set should be used for low voltage contingency violation reporting.

Bus Contingency High Rate Set: This field value indicates which bus voltage rating set should be used for high voltage contingency violation reporting.

Nomogram Percentage: The percentage to which Simulator's study tools will limit a nomogram.

Limiting End: Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

Use Limit Cost: If this field is set to yes, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts, the marginal cost of enforcing the constraint will increase. For setting the piecewise linear limit cost curve, right-click and choose show dialog from the popup menu to access the Limit Group Dialog.

Often monitoring elements according to Area and/or Zone is all that you need. However, if you need to monitor specific groups of power system elements and not others then you need to create some new Limit Groups.

Limit Group Dialog

The Limit Group Dialog reflects much of the same information that is summarized in the [Limit Monitoring Settings](#) Dialog's **Modify/Create Limit Groups** page. The same information can be changed here as on the previously mentioned display. One important feature that is unique to this dialog is the ability to define a piecewise linear limit cost curve, which can be used for setting up "soft" constraints used in the [Optimal Power Flow](#).

To access this dialog, right-click on one of the limit groups defined on the **Modify/Create Limit Groups** page of the Limit Monitoring Settings dialog and choose **Show Dialog**.

Limit Group Name

Identifies the name of the Limit Group that is currently selected. Select a different limit group from the dropdown to modify parameters for that group.

Disabled

Check this option to ignore all power system element limits in the Limit Group. Leave the box unchecked to monitor limits according to the settings of the Limit Group.

Add New Limit Group

Clicking this button will create a new limit group, displaying a dialog in which the name for the new limit group can be specified.

Rename Limit Group

Clicking this button will display a dialog in which a new name for the current limit group can be specified.

Lines/Interfaces

% of Limit for Reporting (Branches)

The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

Line/Transformer Rating Set

You may define eight different ratings to transmission lines or transformers (for more information see the [Branch Information Dialog](#) or [Branch Options dialog](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

Line/Transformer Contingency Rating Set

This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

Treat Transmission Line Limits as Equivalent Amps

Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$AmpRating = \frac{MVARating}{\sqrt{3} * BaseKV} * 1000$$

When doing contingency analysis and the Power Flow Solution Options are set to [Use the DC Approximation in Power Flow](#), this option will be ignored. Line limits will always be reported in MVA when using the DC approximation.

Limiting End of Line

Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

% of Limit for Reporting (Interfaces)

The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

Interface Rating Set

You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

Interface Contingency Rating Set

This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

% of Limit for Reporting (Nomograms)

The percentage to which Simulator's study tools will limit a nomogram.

Use Limit Cost

If this field is checked, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts, the marginal cost of enforcing the constraint will increase.

Once the box is checked, the table for defining the limit cost function will become enabled. The starting point (% Flow) must be at or above 100%. You can then begin inserting additional points in the piecewise linear curve by right-clicking in the table and selecting Insert Point from the popup menu, followed by entering the new percent flow and marginal cost. Note that the cost function must be strictly increasing, meaning the next marginal cost value must be equal to or greater than the immediately previous value.

When constraints are given the ability to use this limit cost curve, it effectively gives the optimal power flow the ability to "dispatch" the limit of the elements according to the marginal costs of the limit cost curve. You are determining the point on the limit cost curve where the shadow price of enforcing the constraint is met. Thus the OPF routine will determine how much you will allow the element to be overloaded by giving the element the piecewise linear curve. The OPF will solve the problem, and optimize the constraints as it determines shadow prices for each constraint and adjusts the limit accordingly during the iterations of the routine. Since these are somewhat flexible limit assignments to the elements, they are sometimes considered "soft" constraints.

Buses/Summary

Low Per Unit Limit

Buses will be flagged as violated if they fall below this per unit voltage.

High Per Unit Limit

Buses will be flagged as violated if they fall below this per unit voltage.

Low-voltage solution flag

Enter a per unit value at which you wish to measure a voltage and consider the solution to be a low voltage solution.

Include Out-Of-Service Buses

Check this check-box to include out-of-service buses.

Bus Low Rating Set and Bus High Rating Set

Individual buses may either take their limit values from the Low/High Per Unit Limits specified, or they may use their own bus-specific limits. Buses that have their own bus-specific limits defined will use the rating sets specified with these settings. In order for the bus-specific limits to be used for a particular bus, the **Use Bus-Specific** field for the bus must be set to YES.

Elements using Limit Group

Lists the number of elements that are members of the selected limit group.

Contingency Options

Contingency analysis options allow the monitoring of elements based on how they are impacted by a contingency relative to base case flows or voltages. Options are available to **Never report violations if the** change due to the contingency is not great enough or **Always report as a violation if the** change is considered significant enough. These options are also available with Limit Group specific settings along with options to **Report as a violation if a bus becomes disconnected** during a contingency and select **How to Monitor Voltage Changes**(either per unit or percentage changes). To use these options as specified with the Limit Group, check the **Use group specific contingency options** checkbox. When running contingency analysis, these settings will override the ones that are specified with the contingency options on the [Advanced Limit Monitoring](#) tab.

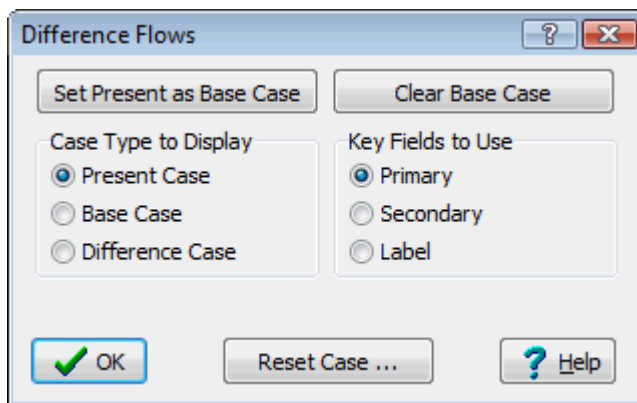
Difference Flows

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, Difference Flows can be used to show the difference in transmission line flows and bus voltages resulting from a contingency or a change in power transfer between two areas. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

For more information about how to use Difference Flows see [Using Difference Flows](#).

The Difference Flows Dialog can be accessed by:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows > Difference Flows** from the [Other Tools Ribbon Group](#).
- Selecting **Difference Flows** from the [online local menu](#).
- All Difference Flows Actions are controlled via the Difference Flows Dialog or the drop-down list on the Difference Flows toolbar button.



Case Type to Display

Use of this feature affects all aspects of the Simulator environment. When using the Difference Flows tool, information shown on online diagrams, case information displays, and [power flow lists](#) is governed by which of the three [Difference Flow Case Types](#) (base, present or difference) is currently being displayed.

Key Field to Use

The difference flows feature must make a comparison between the Present Case and the Base Case to determine which objects between these two case match one another. The Key Fields to Use specifies whether this matching is done using Primary Keys (bus numbers), Secondary Keys (bus names/Nom kV pair), or by Label.

Difference Flows: Case Types

When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three case types is currently being displayed. Difference Flow is available in both [Run Mode](#) and [Edit Mode](#), however when you switch from Run Mode back to Edit Mode, Simulator automatically resets you to the Present Case. This is done so that you do not accidentally try to edit data while in Difference or Base Case mode. Once you are in Edit Mode however, you can still manually switch back to Difference or Base Case Mode.

The case types available are:

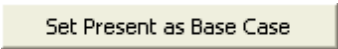
- **Base Case** – A solved power system that serves as the reference for the difference flows tool. To establish a base case, set up a solved power system corresponding to the desired operating point. Open the Difference Flows Dialog and click the button labeled **Set Present as Base Case**.
- **Present Case** – The operating point used in the Difference Flows comparison. **Note:** The Present Case must have the same numbering scheme as the Base Case for proper operation of the Difference Flows tool. See [Topological Differences](#) for more information.
- **Difference Case** – The difference between the Present Case and the Base Case values. The values displayed in the Difference Case are established using the Base Case as the reference. When showing status fields for branches, generators, etc... fields will show up as either OPEN/CLOSED or CLOSED/OPEN if the status changed. The status in the present case will be listed first. A similar format will be used for fields such as generator AGC Status and other text fields.

To toggle between the different case views, open the [Difference Flows Dialog](#) and select the desired case type. Alternatively, you can click the **Difference Flows** button in the [Other Tools Ribbon Group](#) on the [Tools](#) ribbon tab and select the desired case type.

Using Difference Flows

To use the Difference Flows tool:

- Set up a solved power system corresponding to a desired operating point. This operating point will be defined as the [Base Case](#).
- Select **Difference Flows** from the Run Mode ribbon group on the [Tools](#) ribbon tab, or select **Difference Flows** from the [online local menu](#) in Run Mode.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the Base Case.



- Define the operating point ([Present Case](#)) for which to perform the difference flows comparison. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case selecting **Open Case** from the [Application Button](#). In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- See [Difference Flow Case Types](#) for information on toggling between case views. The currently displayed case type is shown in the [PowerWorld Simulator Status Bar](#). When viewing either the Base Case or Difference Case, the corresponding status bar display will be highlighted.

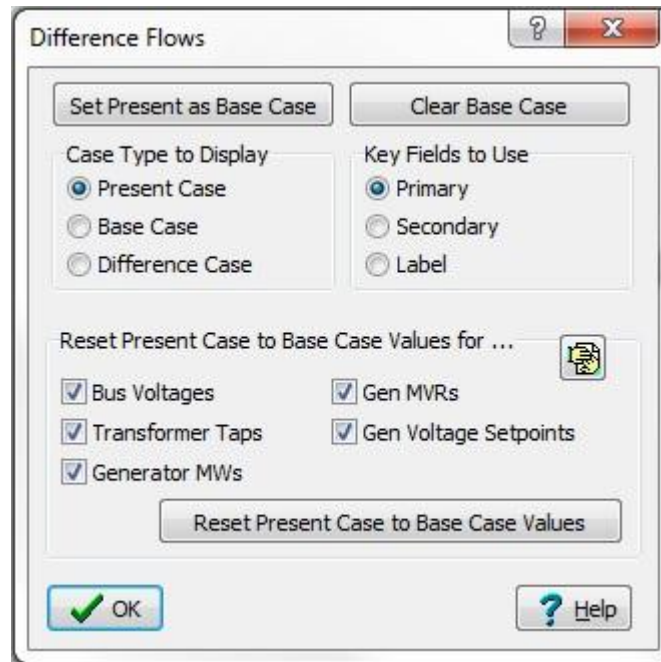


Note that the Difference Flows tool can only be used in Run Mode and the status bar will not display a case type while in Edit Mode.

When showing the Difference Case, most of the fields shown on the onelines and case information displays show the difference between the present value and its Base Case value. For example, on the Generator Records Display, an entry of 0.0 in the MW field indicates that the real power output of the generator did not change. An entry of 10.0 in the MW field indicates that the present real power output of the generator is 10 MW greater than it was in the Base Case.

At any time during a simulation, you can set the present case as the Base Case by clicking the corresponding button on the Difference Flows Dialog.

Conversely, if you have made changes to the present case, and you wish to revert to some or all of the base case values, you can click on the **Reset Case...** button. When you click this button, the dialog will expand to show you options for resetting specific types of values to their base case values.



Expanded Difference Flows Dialog

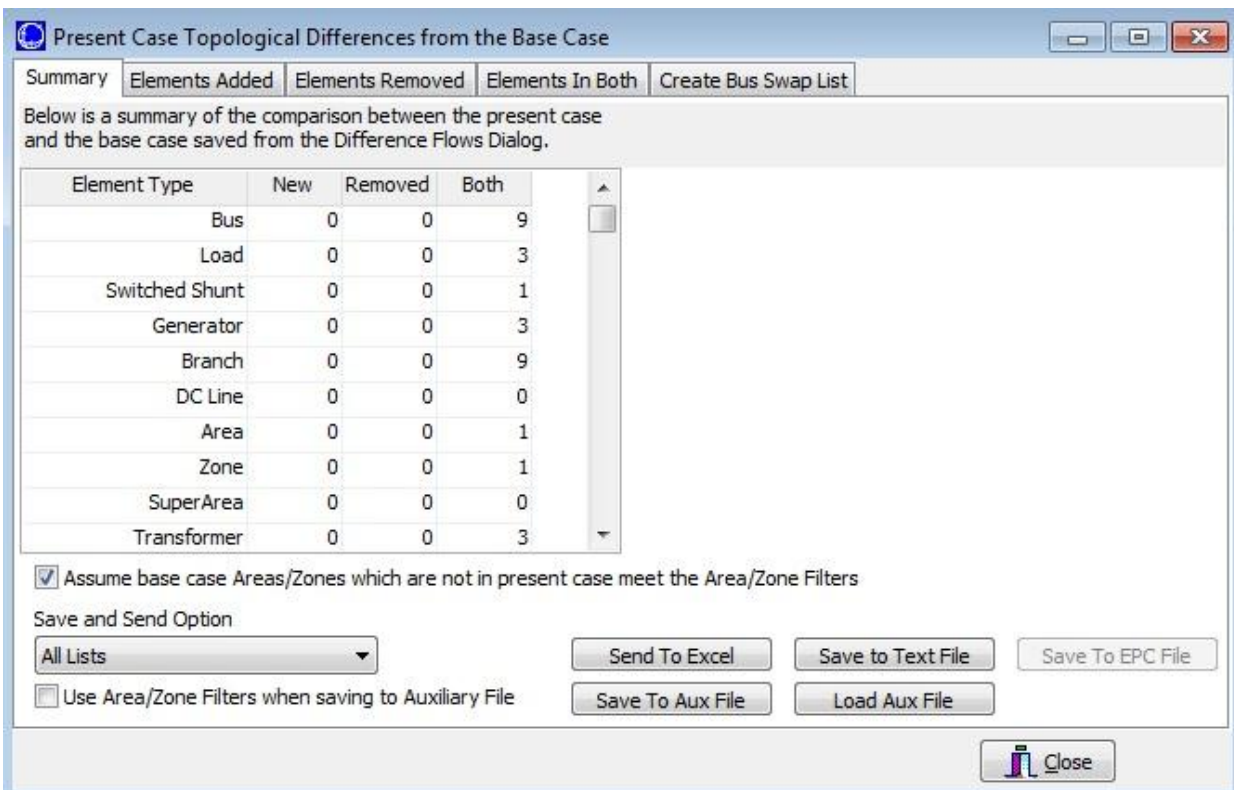
Once you have indicated which types of values you want to reset (by default, all are selected), you can click on the **Reset Present Case to Base Case Values** button to complete the process of reverting to base case values. Note: once you have finished resetting to the base case values, you can hide the Reset options by clicking the Hide Reset Button in the upper right hand corner of the Reset options panel.

The oneline diagrams and case information displays cannot indicate structural differences in the case very well, such as the addition or removal of a device. To identify such differences, make use of the [Present Topological Differences from Base Case](#) tool to identify topology differences.

Present Topological Differences from Base Case

The Present Topological Differences from Base Case option provides the users a way to compare the topological differences between two difference cases in Simulator.

To compare two cases topologically, you must load the first case you wish to use as the reference into Simulator. Use the [Difference Flows](#) tool to set the case as the base case in memory. Once the reference case has been stored as the base case, open the second or comparison case into Simulator. Now if you check the [Tools](#) ribbon tab, and choose the **Difference Flows Menu** on the [Other Tools Ribbon Group](#), you will see the option **Present Topological Differences from Base Case**. Choosing this option will open the dialog to display the topological differences.



Topological Differences Dialog

The Topological Differences Dialog contains five tabs. All but the Summary tab are instances of Case Information Displays and thus exhibit characteristics and controls similar to other displays.

Summary

A listing of the types of objects in the case. The first column displays the number of NEW devices in the comparison case that were not in the base case. Alternately, the Removed column displays the number of devices in the base case that do NOT exist in the comparison case. The third column simply displays the number of items that were matched between the two cases.

Elements Added

Tabular listings of all objects that exist in the comparison case, but not in the reference case.

Elements Removed

Tabular listings of all objects that exist in the reference case, but are not present in the comparison case. A right-click option from the pop-up menu on the Elements Removed tables allows for saving the removed elements selected to a GE EPC file format.

Elements in Both

Tabular listings of all objects present in both cases.

Create Bus Swap List

Used for setting up a bus renumbering list. It is possible that discrepancies in topology between the comparison case and the reference case can be due to a difference in bus numbering between the two cases. [Renumbering the buses](#) in the comparison case may take care of most of the topology discrepancies that are reported in this instance.

Data Objects and Fields

The Save and Send Options on the topological difference dialog makes it possible to save Elements Added, Elements Removed, and Elements In Both to file. In addition to these options, special objects and an object field are available to make accessing these objects easier using script actions and will allow the user to customize the fields stored in the file. When the removed elements are saved in the GE EPC file format the elements are saved with a status value of -4.

To provide easier access to the objects that have been identified as being removed when viewing the topological differences, special object types have been created that allow these objects to be accessed via script commands. These objects provide access to the Elements Removed. All of the object names start with *Removed* with the rest of the name just being the usual object name. To identify buses use the object name *RemovedBus*. To get generators use *RemovedGen*, etc. These objects can be accessed through the [SaveData](#) script command and [GetParametersMultipleElement](#) through SimAuto. These object types can only be stored to an auxiliary file. They cannot be read back in, but, the file headers and object types can be modified to make use of the data or the data can simply be used as documentation of case differences.

A field is available for all object types that can be used with the difference flows tool. This field is found under **Difference Flow\In Diff. Flows Base Case?** and indicated whether or not an object was in the difference flows base case. This field is useful for filtering only the new elements that are only in the present case, field value = NO, or all of the elements in both the present and base case, field value = YES. This will make it possible to use script actions or filters to return only the Elements Added or Elements In Both.

Scaling

Use the **Power System Scaling Dialog** to scale the load, generation, or bus shunts uniformly for either the entire case or a group of selected buses. This display allows you to scale any of the following values:

- Bus real power load
- Bus reactive power load
- Generator real power output
- Real component of the bus shunt admittance
- Capacitive component of the bus shunt admittance
- Reactive component of the bus shunt admittance

To display the Power System Scaling Dialog, go to the [Tools](#) ribbon tab, and choose **Scale Case** from the [Other Tools](#) ribbon group. When the dialog appears, you may begin to select the buses to be scaled. Buses can be selected individually or in a group by areas or zones. In addition, if you already have specific groups of devices defined as an injection group, you can choose to scale values associated with the injection group.

Scale by Bus

The button on the left labeled **Bus** enables selection of loads, generators and shunts by the bus to which the devices are attached. Selection of the buses can be done individually, by their area grouping, by their zone grouping, or by their super area grouping. Scaling is done on the buses chosen regardless of the area, zone, or super area specification. Options for areas, zones, and super areas are included only to aid in selecting and unselecting buses to be scaled. This is important to keep in mind if loads or generators are assigned to a different area or zone than their terminal bus.

The **Buses Table** lists the name and number of all buses in the system and whether or not each bus will participate in the scaling. Similarly, the **Areas, Zones, and Super Area** tables list the names and numbers of all areas, zones or super areas in the system and whether or not each area will participate in the scaling. Simulator initially assumes that no buses are to be scaled. The Power System Scaling Dialog furnishes a number of ways to select the subset of buses to be scaled:

- Use the **Add to Scaling** fields to enter either a [range of areas, zones and/or buses](#) to scale.
- Use the **Remove from Scaling** fields to enter either a [range of areas, zones and/or buses](#) to omit from the scaling.
- Use the **Select buses using a network cut** button to define a [Network Cut](#) to choose a custom set of buses within a portion of the system to be scaled or omitted from the scaling.
- Use the **Buses, Areas, Zones and Super Areas** tables to change the scaling status of individual buses. Simply double-click on the **Scale** field for a bus, area, zone or super area to toggle its value.
- Click the **Set All to Yes** button to scale the entire case. Click the **Set All to No** button to remove the entire case from scaling.

Changing the scaling for an area, zone or super area changes the scaling status for all *buses* in the grouping. For example, to scale all the buses in a single area, first click **Set All to No**. Then, click on the **Scale** field for the desired area in the Area Table. To scale all buses in an area except for a select few, repeat the above process, but then click on the **Scale** field for the buses not to scale.

As you select the buses to be scaled, the fields in the **Totals for Selected Buses** are updated to indicate the total load, generation, or shunt compensation that will be scaled.

Once you have selected the buses, you can either use the **Scale By** fields to enter a new scaling factor for each of the quantities or use the **New Value** fields to specify a new value directly. If you do not wish to scale a particular type of device, such as bus shunts, simply leave the **Scale By** field as unity.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation and Load**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load, generation, or shunt compensation.

Scale by Area

The button on the left side of the dialog labeled **Area** enables the selection of loads, generators and shunts based on the area designation of the device itself. Loads, generators and shunts can have a different area designation than the terminal bus to which the device is attached. In these cases, it is sometimes necessary to use the **Scale by Area** option to choose only the devices that are designated within a certain area, instead of all devices at a particular bus.

To specify the devices within certain areas to be scaled, toggle the **Scale** property of the areas desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of area numbers to be included or excluded from the scaling.

Scale by Zone

The button on the left side of the dialog labeled **Zone** enables the selection of loads, generators and shunts based on the zone designation of the device itself. This is identical to the concept described immediately above in the discussion on **Scale by Area**. As was the case with the area designation, loads, generators and shunts can have a different zone designation than the terminal bus to which the devices are attached.

To specify the devices within certain zones to be scaled, toggle the **Scale** property of the zones desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of zone numbers to be included or excluded from the scaling.

Scale by Injection Group

If injection groups have been defined for the case, you can scale the generation, load, and switched shunt values for the injection group by first clicking on the **Injection Groups** button. The **Buses Table** will be replaced with a new table listing the Injection Groups in the case. To choose the injection groups to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the **Net Injection** fields will update to display the totals for the selected injection groups. When all of the desired injection groups have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected injection groups. Keep in mind that the scaling is done based on net injection and will be done in proportion to the defined [participation points](#).

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control.

Checking the **Scale in Merit Order** option will scale generators in the order of highest to lowest normalized participation factor. This means that each generator participating in the injection group will be moved to either its maximum or minimum limit, depending on the direction of the scaling, before the output of the next generator in the list is changed, until the total scale amount is achieved. If this option is checked, generator MW limits will be enforced regardless of the status of the **Enforce Gen MW Limits** option.

Check the **Scale Only AGCable Generation and Load** option to scale only those generators and loads whose AGC field is set to *YES*. If the **Ignore AGC flag to calculate participation but use AGC flag to scale individual loads or generators** option is checked, the **Scale Only AGCable Generation and Load** is not enabled and will be ignored. If this option is checked, then all generators and loads, regardless of their AGC status, will be used to calculate the participation percentages for all participating elements. Only those generators and loads whose AGC field is set to *YES* will actually be scaled. Normally, if choosing to scale only AGCable generation and load, only those elements whose AGC status is *YES* will be used to calculate the participation percentages.

The choice of **Scale Starting Point** impacts the reference point from which the scaling starts. **Scale from Present Value** scales by adding an incremental change to the present MW values based on the difference between the **New Value** and **Original Value**. **Scale from Zero** sets all participating elements to zero injection (0 MW load or generation) and then scales to the **New Value** from that starting point. When using the **Scale from Zero** option, only a single injection group can be scaled at a time. The following example explains the difference between these two options:

Suppose that Load 1 is 800 MW and Load 2 is 700 MW. These two loads are the only elements in an injection group and they both have a participation factor of 0.5. The desired **New Value** is 1600 MW. Choosing **Scale from Present Value** will result in Load1 being 850 MW and Load 2 being 750 MW. The incremental change in the total load is 100 MW and both loads pick up half of this change. Choosing **Scale from Zero** will result in Load 1 being 800 MW and Load 2 being 800 MW. Both loads have been scaled to 0 MW initially and they each pick up half of the desired 1600 MW total load.

The **Scale Load Field** options provide a way of scaling load that is out-of-service. Selecting **Actual MW (MW Column)** will scale only those loads that are in-service. (The MW column in the [load case information display](#) reflects the total in-service load at a bus.) Selecting **Modeled MW (ignore load and bus status)** will scale all load regardless of the load and bus status.

Click **Do Scaling** to scale the selected injection groups.

Scale by Owners

You can scale the generation and load values by ownership by first clicking on the **Owners** button. The **Buses Table** will be replaced with a new table listing the Owners in the case. To choose the owners to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the Generator and Load MW and Mvar fields will update to display the totals for the selected owner. When all of the desired owners have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected owners.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load and generation for the selected owners.

Note: When scaling generators with multiple owners, the scaling will be done on the entire output of these generators, regardless of the fact that not all of the owners have been set to scale. If this is not a desired behavior, you can define multiple generators at the bus, each with 100% ownership.

Path Distances from Bus or Group

To access the Determine Path Distances from Bus or Group dialog go to the [Tools](#) ribbon tab, and choose **Connections > Determine Path Distances to Bus** from the [Other Tools](#) ribbon group. The Determine Path Distances from Bus or Group dialog provides a way to calculate how far electrically every bus in the system is from a particular starting point.

After clicking the **Calculate** button, all buses that are part of the start element will be labeled internally as being a path distance zero. Then a calculation will be done that determines for *each bus* in the system the shortest total path length in getting from each bus to the Start Element. Buses that cannot reach the Start Element will be flagged with a very large distance (by default 10,000). The results of the calculation will then be copied into the Bus Field specified under the Bus Field to Populate option. The results will be displayed in a case information display at the bottom of the dialog.

The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch.Lines to Process

Lines to Process

Regardless of the Distance Measure above, you can choose which branches allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed. Bus Field to Populate

Bus Field to Populate

Choose a field into which the resulting path distances will be copied. It is recommended that you use the custom floating point or integer fields for this purpose.

Shortest Path Between Elements

To access the Shortest Path Between Elements go to the [Tools](#) ribbon tab, and choose **Connections > Determine Shortest Path Between...** from the [Other Tools](#) ribbon group. The Determine Shortest Path Between dialog provides a way to find the shortest electrical pathway between two parts of the power system.

After clicking the **Calculate** button, the shortest path of AC branches will be determined with the first bus being inside the Start Element and the last bus being inside the End Element. Options described below allow the user to choose the Start Element, End Element, Distance Measure for each branch, and branches that are allowed to be traversed. The results will be displayed in a case information display at the bottom of the dialog showing the sequence of buses that are traversed between the start and end elements.

The options on the dialog are described below.

Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

End Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the ending element. For more help on the object choose see the [Find Dialog Basics](#).

Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X - Per unit series reactance.

|Z| - Magnitude of the series impedance (based on the per unit reactance and resistance).

Length - Length field for each branch.

Number of Nodes - Length of 1.0 is used for all branches.

Other - When choosing Other, click the **Find..** button to choose any numeric field of a branch.

Lines to Process

Regardless of the Distance Measure above, you can choose which branches are allowed to be traversed when finding the shortest path.

All - All branches are allowed to be traversed.

Only Closed - Only branches that are presently closed can be traversed.

Filter - Only branches that meet the advanced filter specified can be traversed. Click **Define...** to choose or create an advanced filter.

Selected - Only branches whose Selected? field is set to *YES* can be traversed.

Find Circulating MW or MVar Flows Dialog

To access the Circulating Mvar and MW Flow Cycle Dialog go to the [Tools](#) ribbon tab, and choose **Connections > Find Circulating MW or Mvar Flows...** from the [Other Tools](#) ribbon group.

Circulating power flows are possible in power systems and can lead to unnecessary power losses and voltage drops in the transmission system. The most common example is circulating Mvars caused by transformers with unbalanced tap ratios. This dialog provides a tool to detect and highlight the circulating real and reactive flows in a power system and also provides a metric to rank the severity of these circulating flows.

A *Flow Cycle* is defined as a series of transmission branches with positive flows that can be traversed to form a loop. Normally these types of cycles are not possible in a power system, but certain configurations of transformer tap ratios or phase shifts can result in circulating Mvar or MW flows. The detailed theory behind this tool is completely described in the follow paper: C. Davis, James Weber, and Kyle Johnson, "Circulating MW and Mvar flows in large systems," in North American Power Symposium, Oct. 2009. You may also email PowerWorld for more details.

Dialog Options

Find Cycles

Click this button to recalculate the list of Flow Cycles after making changes in the other options.

Cycle Type to Find

Choose whether to locate circulating Mvars (much more common) or circulating MWs. Changing this option will automatically recalculate the list of Flow Cycle

Flow Threshold

When searching for flow cycles, enter a threshold below which flows will be ignored. This is needed to remove trivial flow cycles which are below the solution tolerance and are just generally of little interest.

Maximum Related Cycles

It is possible (and quite common) for many *related* cycles to be found in one power system. A group of *related* cycles is defined as a group of cycles which can all be mutually reached by from one another by traversing a series of positive branch flows. (Mathematicians would call this grouping a *Strongly Connection Component* of a *directed graph*). Groups of cycles which can be reach from one another can be combined in many ways to form many new cycles. To prevent the results in Simulator from displaying millions of cycles, this value is entered limit the number of related cycles which will be returned by entering this value. Simulator will return the cycles which have the largest minimum flow.

After finding the Flow Cycles, the lists at the bottom of the dialog show the flow cycle results. The top list shows the list of **Flow Cycles**. As you click on a particular flow cycle, the bottom list will show the **Branches in Selected Flow Cycle** (or Buses). There are also tabs which show **Branches in any Cycle** and **Buses in Any Cycle**. You can use these case information displays to mark all the branches that are in a cycle and then use a visualization tool such as [Dynamic Formatting](#) or [Contouring](#) to show visually show the Flow Cycles on a one-line diagram. Example pictures of this are shown at the bottom of this topic.

Mvar and MW Flow Cycle Dialog

Find Cycles Cycle Type to Find: ☒ Mvar Cycles Flow Threshold: 0.10 Mvar
☐ MW Cycles Maximum Related Cycles: 20

Filter: Advance Flow Cycle Find... Remove

	Related Cycle Group	Area Names	Loss Mvar Reduction	Loss MW Reduction	Cycle X-Weighted Ave.	Cycle Min. Flow	Lines in Cycle	Min Bus Num	Max Bus Num	Min Tap Ratio	Max Tap Ratio	Max Percent MVA Flow
7	7	NYISO	14.06	0.72	85.91	42.64	7	130755	130840	0.90625	1.05346	49.95
8	2	IESO, NYISO	13.96	0.39	194.23	142.44	6	147834	157063	0.91667	1.04545	83.50
9	2	IESO, NYISO	13.96	0.39	194.22	142.44	6	147834	157063	0.91667	1.04545	83.50
10	2	IESO, NYISO	13.71	0.39	175.18	112.69	7	147834	157063	0.91667	1.04545	83.50
11	2	IESO, NYISO	13.71	0.39	175.18	112.69	7	147834	157063	0.91667	1.04545	83.50
12	2	IESO, NYISO	13.71	0.39	175.18	112.69	7	147834	157063	0.91667	1.04545	83.50

Branches in Selected Flow Cycle Buses in Selected Cycle

	Near Bus Number	Near Bus Name	Near Bus Nom kV	Far Bus Number	Far Bus Name	Far Bus Nom kV	Circuit	Cycle Flow	Cycle Flow Direction	Loss MW Reduction	Loss Mvar Reduction	Tap Ratio	Phase (Deg)
1	157051	BECK B	345.0	147834	NIAG 345	345.0	1	201.35	From-To	0.00	0.13	1.00000	0.00
2	147834	NIAG 345	345.0	147841	NIAGAR2E	230.0	1	112.69	From-To	0.03	1.40	0.96500	0.00
3	147841	NIAGAR2E	230.0	147842	NIAGAR2W	230.0	1	126.68	To-From	0.00	0.00	1.00000	0.00
4	147842	NIAGAR2W	230.0	157063	BCK2PA27	230.0	1	190.71	To-From	0.24	2.28	1.00000	0.00
5	157063	BCK2PA27	230.0	157052	BECK2 DK	220.0	27	195.86	From-To	0.07	4.93	1.04545	0.00
6	157052	BECK2 DK	220.0	157059	BECK2PA2	220.0	1	231.02	To-From	0.00	0.07	1.00000	0.00
7	157059	BECK2PA2	220.0	157051	BECK B	345.0	02	231.05	From-To	0.03	4.89	0.91667	0.00

Close

Flow Cycle List Display Results

The Flow Cycle List has the following columns by default.

Related Cycle Group

Related Cycle Group Number will be the same for that which can be reached from one another. These cycles are likely related to one another.

Area Names

Comma delimited list of area names which are traversed by the cycle.

Loss Mvar Reduction, Loss MW Reduction

Estimate of the reduction in Mvar (or MW) losses in the entire cycle if the circulating flow is removed. The larger this number is, the more severe the circulating flow is. The list is sorted by the Loss Mvar Reduction by default.

Cycle X-Weighted Average

Shows the weighted average flow in the cycle weighted by the series reactance (X) values.

Cycle Min Flow

Shows the minimum flow on any branch in the cycle. The larger this number is, the more severe the circulating flow is.

Lines in Cycle

Number of lines (branches) in the cycle.

Min Bus Num, Max Bus Num

Shows the Minimum and Maximum bus number traversed in the cycle

Min Tap Ratio, Max Tap Ratio

Show the minimum and maximum tap ratio on any branch in the cycle.

Max Percentage MVA Flow

Shows the maximum percent MVA flow on any branch in the cycle.

Branches in Selected Flow Cycle

The Branches in Selected Flow Cycle list has the following columns by default.

Cycle Flow

Magnitude of branch's flow for the cycle (the minimum of the from and to flow).

Cycle Flow Direction

Direction of flow in cycle relative to line's direction

Loss Mvar Reduction, Loss MW Reduction

Estimate of the reduction in MW and Mvar losses on this branch if the circulating flow is removed

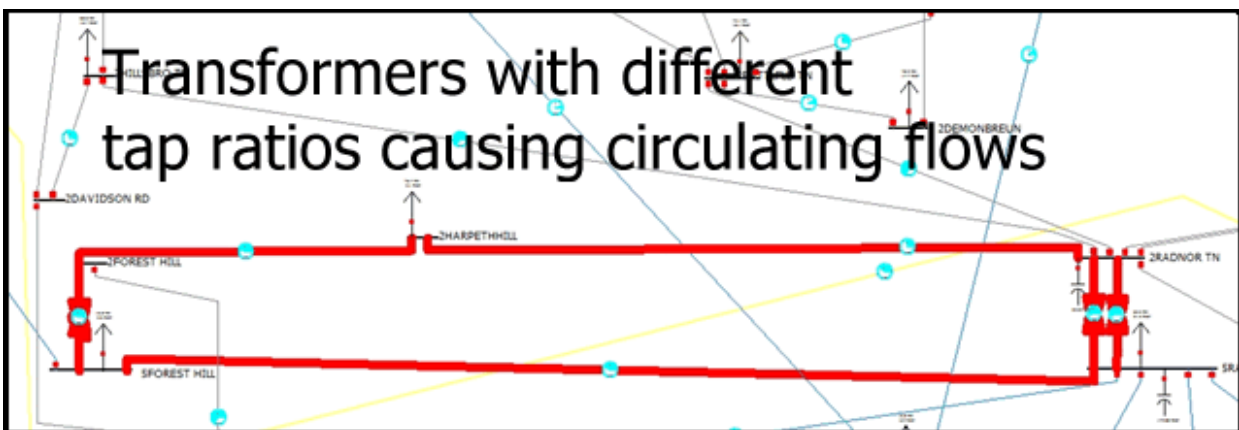
Tap Ratio

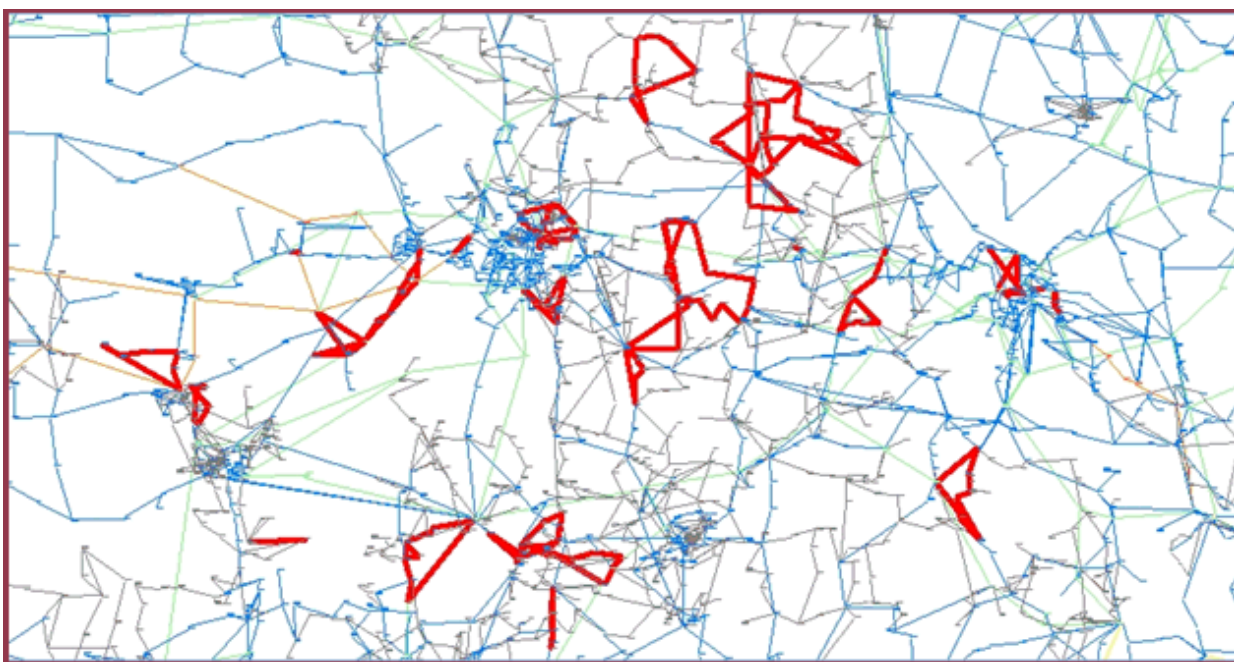
Tap Ratio of the branch

Phase (Deg)

Phase Shift of the branch

Example Visualizations show circulating Mvar Flows





Find Branches that Create Islands

To access the Find Branches that Create Islands, go to the [Tools](#) ribbon tab, and choose **Connections > Find Branches that Create Islands** from the [Other Tools](#) ribbon group.

There are several options for selecting which ac lines to process.

Line Processing Options

All ac Lines

All ac lines in the power system model will be processed.

Use Area/Zone/Owner Filter

All ac lines that meet the defined Area/Zone/Owner Filters will be processed.

Select Area/Zone... to display the Area/Zone/Owner Filters dialog.

Use Selected

All ac lines that have the Selected? field set to 'YES' will be processed.

Click Select Lines... to display all ac lines and change the Selected? field.

Meets Filter

All ac lines that meet a selected advanced filter will be processed.

Use the drop down box to select a defined [advanced filter](#) or click Define Filter to display the Advanced Filters for Branch Dialog. This dialog will allow you to define a new advanced filter for a branch or update an existing filter.

By checking **Do not display radial lines creating a single bus island** those lines that only island a single bus will not be displayed with the results.

Click **Determine Branches** to start the processing once all options have been set.

Because the processing of ac lines in a large power system may take some time, there is an **Abort** button that is enabled once the line processing has started that will stop the processing at any point.

The list of resulting ac lines that create islands will be displayed under **Branches that Create Islands**. To show the list of buses that are islanded by an outage of any line in the list, select a line and the **Islanded Buses** list will be populated. Both the Branches that Create Islands list and the Islanded Buses list are [case information displays](#) and have the same [local menu options](#) and characteristics of other [case information displays](#).

Overview of Facility Analysis in PowerWorld Simulator

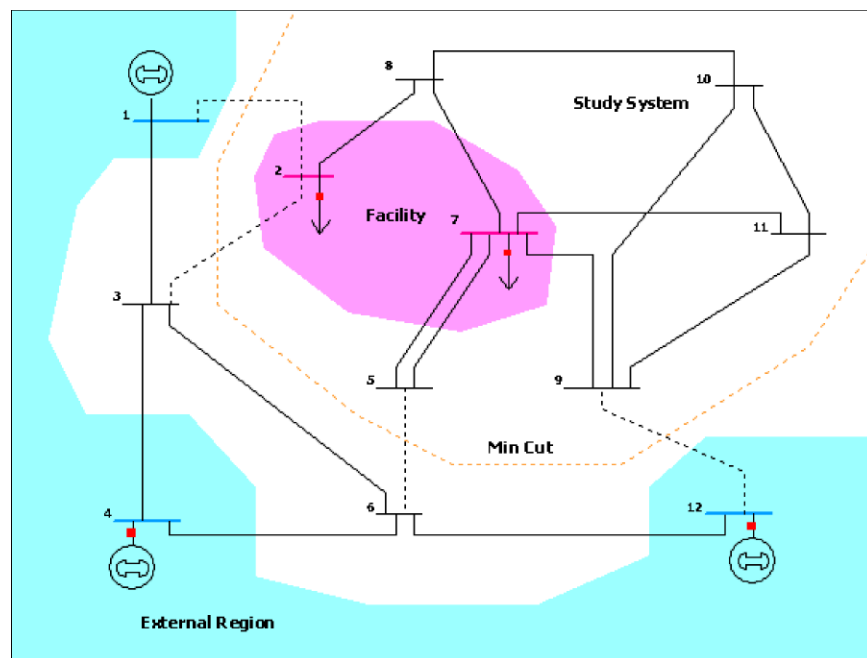
Facility Analysis is used to study the topological redundancy of interconnect-specific electric facilities. This application determines the minimum set of AC transmission lines and transformers that, when opened or removed from the system, would electrically isolate a set of Facility buses from a set of External buses.

The tool is an application of the [augmenting path max flow min cut algorithm](#) with modifications to handle electric networks.

The Facility analysis process has two steps:

1. The [Select the Buses](#) dialog is used to specify the External and Facility buses. Multiple selections of the External buses can be done using any of the area or zone selectors. The Facility buses are specified by setting the **Selected** field of buses to **YES**.
2. The [Facility Analysis dialog](#) is used to determine the Min Cut and visualize the branches that belong to the min cut.

The Facility analysis application runs in Edit Mode and takes into consideration the open or closed status of the branches. Open lines are considered as not present in the system. The figure below illustrates the functionality of the Facility Analysis tool.



Facility Analysis Dialog

This dialog is used to determine the branches that would isolate the Facility from the External region as specified in the [Select the Buses](#) dialog. When switching to the Facility Analysis page, the application builds a graph data structure and reports information regarding the External Region and the Facility.

To access the Facility Analysis Dialog go to the [Tools](#) ribbon tab, and choose **Connections > Facility Analysis Dialog** from the [Other Tools](#) ribbon group.

Select the Buses

The options and use of this page are described in their entirety in the topic titled [Bus Selection Page](#).

Facility Analysis

External Region

The external region is a set of buses from which the Facility would be isolated. Although the buses in the external region may not be adjacent to each other, the algorithm will assume that any of these buses can supply electricity to the Facility and thus the buses in the External region are considered to be connected. The External region is defined using the [Select the Buses](#) dialog.

External Region - Number of Buses

Indicates the number of buses in the External region. This number is equal to the number of buses in the system that were labeled as External in the [Select the Buses](#) dialog. There should be at least one bus in the external region.

External Region - Capacity

Indicates the number of branches that connect the external buses to study buses in the system. This is the outgoing [graph flow](#) capacity from the External region toward the Facility.

Facility

Is the set of buses that constitute the Facility. Although the buses in the Facility may not be adjacent to each other, the algorithm will assume that any of these buses may receive power from the External region, and thus the Facility buses are considered to be connected to each other.

Facility - Number of Buses

Indicates the number of buses in the Facility. This number is equal to the number of buses in the system that were labeled as *Study* and whose **Selected** field is set to *YES*. There should be at least one bus in the Facility.

Note that if a bus was specified to be *External* and its **Selected** field is *YES*, then the application will issue an error, since Facility buses cannot be in the External region. In such cases the status will indicate that the graph structure is incomplete.

Facility - Capacity

Indicates the number of branches that connect Facility buses with study buses. This number is equal to the outgoing [graph flow](#) capacity from the Facility toward the External region.

Status

Shows the status of the Facility Analysis application. If the External region and the Facility are specified, the status will indicate that the graph structure is ready. During execution of the Min Cut algorithm, the status shows the number of the augmenting paths found so far and the number of branches in the current path.

Find Minimum Cut

Press this button to initiate the [Min Cut algorithm](#). The button is inactive if the graph structure is incomplete, i.e., there is either no external bus or facility bus, or facility buses were found inside the External region.

Show Paths

Toggle this button to visualize the augmenting paths at the bottom of the form. The augmenting paths are listed in the order in which they were found. Note that the first path has fewer nodes, since the algorithm uses a shortest (least number of nodes) path routine. For each augmenting path the number of buses in the path, as well as the corresponding bus numbers are listed in the dialog. This button is enabled only if a min cut has been found.

Augmenting Path Max Flow Min Cut Algorithm

The augmenting path max flow – min cut algorithm is used to identify the minimum number of branches that need to be opened or removed from the system in order to isolate the Facility (power system device) from an External region.

The algorithm is an application of the Max Flow - Min Cut theorem, which states that the maximum flow that can be transferred from a set of source nodes to a set of sink nodes across a graph equals the capacity of the minimum cut. The facility analysis in Simulator finds a min cut, although this cut may not be unique.

The application consists of three stages:

1. **Convert the electric network to a graph structure**

In this stage, each branch of the system is converted to an undirected arc with [graph flow](#) capacity equal to one. Thus, only one unit of [graph flow](#) can be sent through a branch. The Facility buses and the External buses are converted to Facility and External supernodes, respectively. This effectively reduces the problem to finding the min cut between these two supernodes. The number of nodes and capacities of the Facility and the External region are also computed during this stage.

2. **Find the Max Flow using the Augmenting Path Algorithm**

This is an iterative process. At each step, the algorithm finds a new augmenting path from the Facility to the External region and augments the [graph flow](#) along this path in one unit. Consequently, the branches in the path won't be available for flow augmentation in the next step. Each new path is determined using a shortest path routine. The algorithm stops when no augmenting path from the Facility to the External region can be found. The number of units transferred from the Facility to the External region (path augmentations) reaches the number of branches in a certain cut. Note that the number of branches in the min cut can not exceed the capacity of either the Facility or the External region.

3. **Determine the branches in the min cut**

The identification of the branches that constitute the min cut consists in tracking down labels in the buses and branches used during each path augmentation.

Graph Flow

Most network and graph theory applications use the concept of flow to represent any object that can be transported, such as communication packets or trucks, but also connectivity properties of graphs. In the [augmenting path max flow min cut algorithm](#) the flow is an artificial concept used to represent topological connectivity of buses. Two buses are adjacent if flow can be sent from one to the other through a branch.

Graph Flow Capacity of a Branch

Networks that transport some flow are said to be capacitated if its arcs (here synonym of "branches") have some limit associated to the flow transportation. For instance, capacity of a communication channel, or number of trucks that can be simultaneously on a certain road. The algorithm used in the Facility Analysis assigns a capacity of one to each branch. This means that the branch can be used only once for "connecting" two nodes.

Graph Flow Capacity of a Cut

The capacity of a topological cut is equal to the sum of the capacities of its arcs. In the Facility Analysis, the capacity of the min cut is equal to the number of branches in the min cut, since each branch has a capacity of one.

Governor Power Flow

The Governor Power Flow dialog shows all information related to solving a governor power flow. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. Using this dialog, you can modify settings related to solving the case while on Governor or "Island-Based Automatic Generation Control (AGC)".

The Governor Power Flow Dialog has two tabs, Options and Generator Options, which are described below.

Generator Options Tab

This tab sheet presents a case information display with all the generators of the case. The columns shown in the display are helpful when looking at solving a Governor Power Flow.

Options Tab

This section describes the options available on the Options tab of the Governor Power Flow dialog. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab.

This tab sheet presents the following controls:

Disable Automatic Generation Control

Disables automatic MW generation changes during the power flow solution.

Island AGC Tolerance

The ACE mismatch tolerance allowed for the Island to be considered solved in terms of generation dispatch.

Island-Based Automatic Generation Control

Select how the generations should be controlled for the dispatch.

- Disabled: The area and super area dispatch settings from the case will be used
- Use Participation Factors of individual generators: Each generator will have it's own participation factor, and will contribute according to its participation factor divided by the sum of all participation factors of all other generators in the same island.
- Calculate Participation Factors from Area Make Up Power Values: Each area is assigned a "factor" as to how it should participate towards the generation dispatch in the island. This is similar to participation factors for generators. The total percentage the area contributes towards the generation change needed in the island is equal to its individual factor divided by the sum of all area make up power factors. Then within each area, the generator participation factors determine how the area's percentage is made up of available generation within the area. The area make up power values can be set in the table of areas on the right hand side of the dialog.
- Dispatch using an Injection Group: The island dispatch will be made up by a combination of generators and loads, defined in an [injection group](#) that can be selected from the dropdown list.

How should reactive power load change as real power load is ramped?

If you are using an injection group with load as part of the dispatch, then you can specify how the reactive power load should respond as the real power demand of loads changes with the dispatch. The reactive power can either be kept at the starting ratio of real and reactive power of the load, or the MVAR amount can change at each load by a specified power factor.

Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab

This tab sheet presents a case information display with all the generators of the case. The columns shown in the display are helpful when looking at solving a Governor Power Flow.

Governor Power Flow: Options Tab

This section describes the options available on the Options tab of the Governor Power Flow dialog. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab.

This tab sheet presents the following controls:

Disable Automatic Generation Control

Disables automatic MW generation changes during the power flow solution.

Island AGC Tolerance

The ACE mismatch tolerance allowed for the Island to be considered solved in terms of generation dispatch.

Island-Based Automatic Generation Control

Select how the generations should be controlled for the dispatch.

- Disabled: The area and super area dispatch settings from the case will be used
- Use Participation Factors of individual generators: Each generator will have its own participation factor, and will contribute according to its participation factor divided by the sum of all participation factors of all other generators in the same island.
- Calculate Participation Factors from Area Make Up Power Values: Each area is assigned a "factor" as to how it should participate towards the generation dispatch in the island. This is similar to participation factors for generators. The total percentage the area contributes towards the generation change needed in the island is equal to its individual factor divided by the sum of all area make up power factors. Then within each area, the generator participation factors determine how the area's percentage is made up of available generation within the area. The area make up power values can be set in the table of areas on the right hand side of the dialog.
- Dispatch using an Injection Group: The island dispatch will be made up by a combination of generators and loads, defined in an [injection group](#) that can be selected from the dropdown list.

How should reactive power load change as real power load is ramped?

If you are using an injection group with load as part of the dispatch, then you can specify how the reactive power load should respond as the real power demand of loads changes with the dispatch. The reactive power can either be kept at the starting ratio of real and reactive power of the load, or the MVAR amount can change at each load by a specified power factor.

Set Generator Participation Factors

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its area's load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

Max MW Rating of Generator The participation factor for each generator is set to the generator's maximum MW capability.

Difference Between Max and Current Output The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

Constant Value of The participation factor for each generator is set to the same hard-coded value.

File The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check

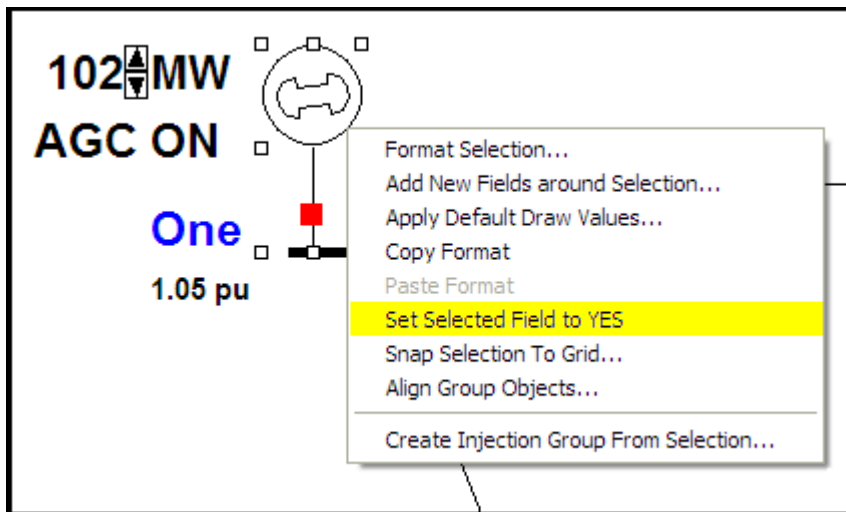
the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

Set Selected Field

The Set Selected Field option is an often unused featured in Simulator, but can at times prove very useful. Each object in Simulator has a property called **Selected?**, which does nothing in Simulator other than allow the user to choose a specialized set of objects for some other purpose, such as [advanced filtering](#) a display. The user can add a column to most types of [case information displays](#) which shows the value of the **Selected** property for the type of object being observed. By default, this field is always set to *No*. However, the user can change the value of this field, and then sort the column, filter the display, or any other action that can normally be performed on a column of a Yes/No type.

Set Selected Field to YES from a Oneline diagram

If you right-click on a group of selected oneline objects, one of the local menu options that appears is Set Selected Field to YES. Choose this option to change the Selected field for all the underlying data objects' Selected field to YES.



Set Selected Field for Network Cut

In addition to modifying the **Selected** property manually in a [case information display](#), it is also possible to define a group of buses' **Selected** property by defining a [network cut](#) on the system. Choosing the **Set Selected Field for Network Cut** option from the **Tools** menu will open the [network cut](#) dialog automatically for setting the **Selected** property of a group of objects according to a desired cut plane chosen.

Network Cut

The Network Cut tool is another method for choosing sets of buses for such features as [Scaling](#) or [Equivalencing](#).

The use of the network cut method is to define a set of branches as the "cut" plane, then choose a bus on one side or the other of the cut to indicate which side of the cut you are interested in. The network cut dialog provides the functionality necessary for defining the network cut.

Defining Network Cut

The first step in defining the network cut is to choose the branches which define the cut plane in the system. The one caution is to be sure you select a closed loop cut plane. In other words, you must select a set of branches which completely topologically separates two portions of the system. If you only choose a partial cut plane, which does not completely cut the system in two distinct pieces, then attempted use of the ill-defined cut plane will fail.

To choose the branches which define the cut plane, simply highlight the branches by clicking on them in the list. For each branch you highlight, you must click on arrow button (pointing to the box on the right) to add the branch to your cut plane definition. You can make use of the control (Ctrl) and shift keys to select multiple branches at one time in the list of branches. Note that you can also select a cut plane similarly using interfaces or DC lines.

Once you have the branches (or interfaces or DC lines) selected which form the cut plane, you then need to choose a bus on either side of the cut plane to indicate which side you are interested in. The bottom panel on the display allows you to locate and select this bus.

Require Paths to be Energized

This option can be checked if the branches forming a network cut should be energized to be included.

Include How Many Tiers of Neighbors

Once the network cut has been defined by a set of branches and the bus defining which side of the cut is being examined has been chosen, this option will then include buses within so many tiers of the network cut boundary, on the opposite side of the cut as the specified bus. If the number of tiers is set to zero, then the buses examined will only be those on the same side of the cut as the bus selected.

Setting the Field

Once the network cut has been defined and the side of the cut to be examined has been chosen, the appropriate field for each bus in the area of interest can be set. If the network cut is for the equivalencing tool, then the field to set is the **Which System** property, which defines if a bus is in the external or study system when creating an equivalent. If the network cut is for the scaling tool, then the field to set is the **Scale** property, which defines if a bus is to be included in a scaling action.

Filter by KV

The buses within the network cut can be filtered by nominal voltage level before the field(s) are set. Only buses that are within the network cut and are within the nominal KV level specified will have their fields set to the desired value.

Use Area/Zone Filters

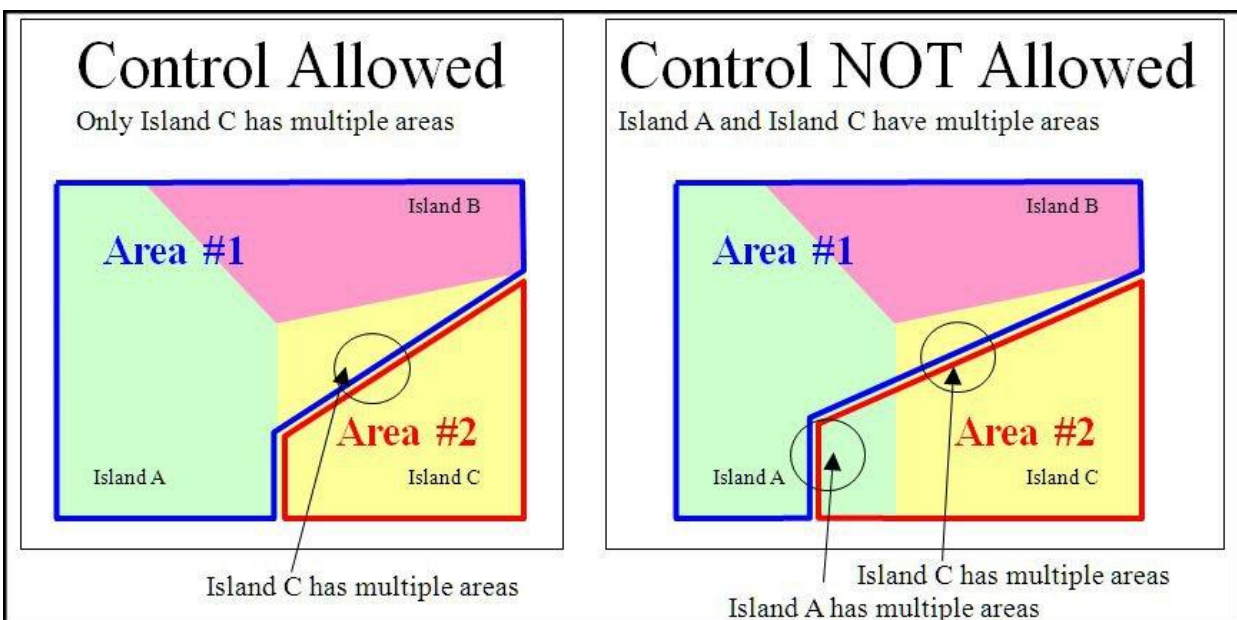
The buses within the network cut can be filtered by the area or zone in which they are contained before the field(s) are set. Only buses that are within the network cut and whose area meets the defined Area/Zone filter specified will have their fields set to the desired value.

Create New Areas for Islands

When an area spans multiple electric islands, area MW interchange control may not be possible.

Consider two scenarios shown below. The scenario on the left has Area #1 that spans three islands while Area #2 is contained within a single island. Island C contains both Area #1 and Area #2. Because Area #1 only spans one island that contains more than one area, area interchange control is allowed. When balancing the area interchange in Island C, it can easily be determined what generators will respond. Only those generators in Area #1 in Island C will respond with interchange between Area #2. Internally when the power flow solution is solved, Simulator creates temporary areas for the portion of Area #1 that is in Island A and the portion that is in Island B. This ensures that only those generators that are in Island C will respond to interchange with Area #2. In this scenario, both Area #1 and Area #2 are allowed to be on area interchange control.

The scenario on the right has Area #2 that spans two islands and Area #1 that spans three islands. Because both of the areas are contained in islands that have more than one area, area interchange control is not allowed for both Area #1 and Area #2. Control is not allowed because Simulator does not have enough information to determine in which islands the area transactions are supposed to occur. In this scenario, Simulator will turn off AGC for areas that meet these conditions.



With the Create New Areas for Islands option, permanent areas can be created that match the areas that Simulator creates temporarily while solving the power flow. New areas will get created if an area is on AGC, spans multiple viable islands, and only one of those islands has more than one area in it. An island is considered viable if it has closed non-zero load and an online generator or it has an online generator that is connected to the rest of the system via a DC line. For an area meeting these criteria, new areas will get created for the portion of the area in islands that contain only one area. If an area spans multiple islands and each island contains only a single area, new areas will get created for the islands that have the fewest buses. New areas will only get created if there are generators on AGC or online load in more than one of the islands in which the area has buses. New areas will use the same AGC method as their original area unless that area is on area slack bus control. In that case, the new

areas will be on no AGC. New areas will be names with the original area name with the slack bus for the island appended.

Browse PWB File Headers

The Browse PWB File Headers is accessed by selecting **Other > Browse PWB File Headers** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab.

When selected, Simulator will prompt you to specify a directory containing PWB files for which you wish to preview the header (case description) text. Once the directory has been specified, Simulator will obtain the case description for each PWB file in the directory, and write them all into the Simulator message log.

Browse Open Onelines

All the objects in all the open .pwd files can be listed in the Browse Oneline Environment dialog. To do so, go to the [Onelines](#) ribbon tab, [Active](#) ribbon group and choose **List Display > Browse open onelines** in Edit Mode or Run Mode. The power system objects are listed on tabs according to their type. For instance, all the buses and bus fields are listed in the **Buses** tab while all the generators and generator fields are listed in the **Generators** tab. The **Others** tab lists all the objects on the onelines which are not associated with power system elements, e.g. background lines or images. The **All** tab lists all the objects in all the open onelines. The default display columns for each grid include the oneline file name of the object and the type of the object. Objects that are linked to case elements also have identifying information about the data element that they represent. The **Others** and **All** pages include the default display columns for generic display objects.

The oneline displays can be panned to the selected object. To do so, select one object from the Browse Oneline Environment dialog and right click to invoke the local popup menu. Click **ScreenObject Records > Pan to Object on Open Onelines**. This will bring the oneline with the selected object to the front and pan to the selected object. When panning, the browsing dialog can be kept on top if **Keep browsing dialog on the top when choosing Pan Oneline to Object** is checked. Otherwise, the dialog will be sent to back when panning.

Unused Bus Numbers

When making changes to a power system case, it is often beneficial to know which bus numbers are still available. To save a list of unused bus numbers to a text file, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Modify Case > List of Unused Bus Numbers** from the [Edit Mode](#) ribbon group. Enter a [range of bus numbers](#) in the dialog and click **OK**. In the Save As Dialog, specify the text file where the list of bus numbers should be written and click **Save**. Only those unused bus numbers in the specified range will be listed in the file. Clicking **Cancel** at any point will abort saving the list to a file.

Equivalents

An equivalent power system is a power system model of smaller dimension than the original system that approximates the behavior of the original system reasonably well. In reality, most power system models are actually an "equivalent" of a much larger interconnected network. When performing power system studies, it may be desirable to reduce the size of the system model even further so that it may be solved more quickly.

To bring up the [Equivalents Display](#), while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Equivalencing...** from the [Edit Mode](#) ribbon group.

The most important part of constructing an equivalent is determining which buses should be explicitly retained in the equivalent, and which buses should be *equivalenced*, or removed from the case. Several definitions are useful here:

Study System

The buses that are to be retained.

External System

The buses that are to be equivalenced.

Boundary Buses

Any buses in the study system that are connected to buses in the external system.

How well the equivalent system approximates the behavior of the original system depends upon which buses are retained in the study system. Retaining more buses yields results that more closely match those of the original case, but at the expense of greater computation time. The number of buses to retain in the study system depends upon how the equivalenced system will be used. Building system equivalents is as much an art as it is a science, with few solid rules of thumb. However, to improve accuracy, you should retain as many generator units as possible.

The actual equivalent is constructed by performing a matrix reduction on the bus admittance matrix. A result of this process is the creation of "equivalent" transmission lines that join boundary buses equipped with equivalent shunts or loads. Equivalent lines typically have a circuit identifier of *99*, but have also been seen to have other numerical values between 90 and 99, or an alphanumeric identifier of *EQ*. Since many of the equivalent lines created during the matrix reduction have very high impedance values, an option is provided to ignore equivalent lines with impedances exceeding a specified threshold value. Additionally, an option is provided to convert the equivalent shunts added at the boundary buses to constant PQ loads. These PQ loads will be given circuit identifiers similar to those given to equivalent transmission lines.

Equivalents Display

The **Equivalents Display** is used to construct equivalent systems. An *Equivalent System* is a system of smaller dimension that exhibits similar power flow properties. Equivalent systems are constructed to help accelerate computation time without sacrificing a significant amount of accuracy. For more information, please see [Equivalents](#).

To bring up the Equivalents Display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Equivalencing...** from the [Edit Mode](#) ribbon group. This display contains two pages, the **Select The Buses Page** and the **Create The Equivalent Page**. Use the Bus Selection Page to partition the power system into the study system and the external system. Use the Create Equivalent Page to

- Save the external system in a file
- Extract the external system
- Build an Equivalent

Each of these tasks is described below.

Select The Buses Page

To perform any of the tasks described on the Create Equivalent Page, you first need to specify the study system and the external system. Do this by directly assigning buses to the desired system. The [Select The Buses Page](#) has been designed to provide a number of powerful and complimentary ways of accomplishing this task.

The most important point to keep in mind when using this page is that membership in the study system and the external system is on a bus-by-bus basis (as opposed to by areas or zones). Thus, each bus is either in the study system or the equivalent system. Each bus' current assignment is indicated in the Buses list, which is shown on the bottom left corner of the page. The Buses list is a [Case Information display](#) and can be manipulated similar to other displays. By default, all buses initially reside in the study system. Please see [Select The Buses Page](#) for more details.

Create The Equivalent Page

The Create Equivalent Page allows you to save the external system in a file, to delete the external system, and to build the power system equivalent.

Build Equivalent

This procedure constructs an equivalent system. For background on equivalents, please see [Equivalents](#). The following sections contain the options for building an equivalent.

Delete All External Generators or Retain Generators with Max MW Ratings Above

If the **Delete All External Generators** checkbox is checked, the equivalencing routine will remove all external generators from the case, regardless of their MW rating. Otherwise, the equivalencing routine will add to the study system any generators originally in the external system whose real power output exceeds the specified value. Retaining large generators often makes an equivalent significantly more accurate. If you do not wish to retain any additional generators, check the **Delete All External Generators** checkbox.

Retain Remotely Regulated Buses

Some generators and transformers regulate buses other than their terminals. When this box is checked, these remotely regulated buses are automatically included in the equivalent if the regulating generator or transformer is included. If the box is not checked, the regulated bus is set to the terminal of the retained object. It is strongly recommended that you leave this box checked at all times.

Retain Branch Terminals For

This section allows you to customize the retention of branch terminals for special-case types of branches. You may choose to retain terminal buses for transformers, zero impedance ties, area tie lines, and/or zone tie lines.

Max Per Unit Impedance for Equivalent Lines

During the equivalencing process, a number of equivalent lines are created joining the boundary buses. All equivalent lines with per unit impedance values above this threshold are ignored.

Two Character Circuit ID for New Equivalent Lines

Choose the circuit identifier to be used for the equivalent lines that are created. Choose between *97*, *98*, *99* or *EQ* for circuit IDs of equivalent lines.

Remove External Objects from Onelines

This feature removes display objects associated with the external system from any open onelines.

Convert Equivalent Shunts to PQ Loads

During the equivalencing process, shunt elements are added at the boundary buses. Check this box if you would like these equivalent shunts converted to constant PQ loads. If this option is checked, equivalent loads are created with a [load ID](#) of *99*.

Remove Radial Systems

Checking this option results in all radial connections in the network to be reduced to their nearest non-radial bus (i.e. node.) The equivalencing routine will iteratively reduce the network when this option is checked, until no more radial connections exist in the system.

Delete Empty Areas/Zones/Substations that occur from Equivalencing

Since equivalencing is a process which ultimately removes buses from the system, and Areas, Zones and Substations are system devices which are groups of buses, this option will automatically remove the definition of these types of objects when all buses within them are removed from the case during the creation of the equivalent.

Adjust Area Unspecified Interchange to Zero Out ACE

Selecting this option will allow Simulator to automatically change the unspecified MW interchange amounts of each area to remove any non-zero ACE discrepancies caused by the equivalencing. This will help prevent these discrepancies from being assigned to the system slack bus.

Include Generator Dynamic Equivalents

Selecting this option will allow Simulator to automatically create dynamic equivalents of generators.

Minimum PU H for New Equivalent Generators

During the equivalencing process, a number of equivalent generators are created. All equivalent generators with per unit impedance values below this threshold are ignored.

Two Character Circuit ID for New Equivalent Generators

Enter the circuit identifier to be used for the equivalent generators that are created. Default is *EQ*.

Select **Build Equivalent System** to construct the equivalent system. Constructing an equivalent system permanently removes the external system from the case and adds a number of equivalent lines and shunts/loads.

Saving the External System in a File

This procedure allows an external system to be saved in a file **without deleting** the external system. This option is useful for allowing you to save a portion of the system in a file, modify it

using perhaps another program, and then use [Append Case](#) to append the modified file to the original case.

When saving the external case, there are two options: 1) save just the external case, or 2) save the external case and any ties to the original case. Option one just saves the external case, while option two saves the external case and any transmission lines or transformers that connect the external system to the rest of the system. Save just the external case if you are planning to use the external case as a standalone case. Save the external case and its ties if you are planning to modify the external case and then to append it back to the original case.

Click **Save External System** to save the external system in a file. You will be prompted for the desired [case format](#).

Deleting the External System

This procedure deletes the external system from the original case. All devices in the external system are removed, including any buses and lines/transformers in the external system and any lines/transformers that join the external system to the study system. Check the **Remove External Objects from Onelines** to remove any display objects linked to the external system from the open onelines.

This command **permanently removes** the external system from the case; an equivalent system is **not** created. Select **Delete External System** to actually delete the system.

Merge Bus Shunts Across Zero Impedance Branches

This procedure merges bus shunts that are across zero-impedance branches.

Merge Bus Shunt Values Above

During the merging process, only will merge bus shunts values above this threshold.

Merge Across Branches

During the merging process, only will merge bus shunts across branches with X, R less than the specified threshold.

Merge Shunts

Press the button to merge the shunt buses as specified by the previous options.

Select The Buses

The Bus Selection page is currently used with two displays:

- The [Equivalencing Display](#), to partition the system into the study subsystem ("Study") and the external subsystem ("External").
- The [Facility Analysis Dialog](#), to determine the buses in the external subsystem from which the buses in the facility would be isolated.

The following description concerns the Equivalencing Display, but the functionality of the Bus Selection Page is similar for both applications.

Buses, Areas, and Zones lists

For both equivalencing and facility analysis, use these lists to manually change the system designation of individual buses, areas, or zones. Note that changing the system designation on the area or zone tab is just another form of changing the designation of study / external system of individual buses. Click on the **Which System?** field in each of these tables to toggle the object's affiliation with the study or external systems, subject to the values of the **Filter by kV** option and the **Include how many tiers of neighbors?** field.

In addition, the bus selection table for [facility analysis](#) contains one additional column labeled **Selected**. This Yes or No field indicates which buses in the Study system belong to the Facility being analyzed. Double-clicking one of the fields in this column will toggle the value of the field between *Yes* and *No*.

External

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the external system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the external system.

Study

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the study system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the study system.

Include how many tiers of neighbors?

This value indicates the number of tiers of neighbors to carry with each selected bus when adding the selected bus to either the study or external system. For example, if Neighbor Tiers is 1 and we elect to add bus X to the external system, both bus X and its first tier of neighbors will be added to the external system. If Neighbor Tiers is 0, only bus X will be added to the external system.

Filter by kV

If the **Filter by kV** box is checked, then only buses having a nominal voltage level between the values given in the **Max kV** and **Min kV** fields can be selected.

Set All As External

Click this button to assign all buses to the external system.

Set Branch Terminals External or Study

Clicking either of these two buttons allows setting advanced filter criteria which define a branch or group of branches. Once the criteria are set, clicking **Filter** will select the terminal buses of all branches meeting the filter criteria, and set those buses to either the External or Study system, depending on which button was pressed.

Select Buses using a Network Cut

A custom [network cut](#) can be defined for choosing which buses should remain in either the Study or External system.

Save Buses to File

Once the system has been partitioned, this command allows you to store the numbers of the buses of the study system in a text file.

Load Buses from File

Click on this button to load a listing of the buses to be included in the study system from a text file. You will be prompted to select the text file. The format of this text file is such that one bus number occupies each line. Any buses not identified in this file are defined as being in the external system.

Example

Assume you would like to create an equivalent containing all the buses in areas 1-5 and 10, plus any tie buses, and bus number 2050.

- Since initially all buses are in the study system, first enter 1-1000 in the **Areas** field of **Add to External System**. Since the area of every bus is within this range, this places all the buses in the external system. Alternatively, click **Set All as External** to accomplish the same objective.
- Set **Include how many tiers of neighbors?** to 1. This indicates that all subsequent selections will affect the specified buses and their first tier of neighbors.
- In the **Areas** field of **Add to Study System**, enter 1-5,10. This places all the buses in these areas, plus any tie buses (since Neighbor Tiers is 1), into the study system.
- In the **Buses** list, double-click in the **Which System** field for bus 2050 to change its status from *External* back to *Study*.

Appending a Case

The Append Case command allows you to append additional power system components to an existing case. Unlike the [Open Case](#) command, Append Case does not delete the existing case (if any) before loading the selected case. To append a case to the existing case, go to the [Tools](#) ribbon tab and choose **Modify Case > Append Case** from the [Edit Mode](#) ribbon group.

The Append Case command can be useful when used in conjunction with the [Equivalencing Display](#).

See the [Case Formats](#) topic for information on special handling of data that occurs when appending a case using specific file formats.

Merging Buses

Two or more buses can be merged to a new bus in Edit Mode with the loads, generators, and shunts of the buses merged moved to the new bus. The transmission lines among the buses merged will be deleted while the transmission lines connecting the merged buses and buses which are not selected to merge will be moved to the new bus.

To join two or more buses together using the oneline display, select at least two power system elements (at least one of them being a bus), then right click on one of the selected elements to invoke the popup menu and select **Merge selected buses**. In the Bus Merging Dialog, specify the buses to merge further if needed by clicking **Add** and/or **Delete** button. To add buses to merge, click **Add** and select all the needed buses in the **Choose a Bus** dialog and click **OK**. To delete from already selected buses, select the buses to delete and click the **Delete** button.

Once you have the elements to merge selected, enter the number, name, nominal voltage, zone number, area number and substation number for the new bus. These properties can be set to be the same as one of buses to merge by selecting it in the **Specify buses to merge** box and pressing the **Set new bus properties same as selected bus**.

Buses can also be merged in the [Bus Case Information Display](#). To do so, select a cell and right click to popup the local menu. Select **Merge selected buses**. The buses to merge and the properties of the new bus can be edited in the Bus Merging Dialog.

When buses are merged from the oneline diagram, the selected buses will be joined in both the PWB case and the oneline. When buses are merged from the bus grid, the selected buses will be joined only in the PWB case but not in the oneline. This might result in bus objects not connected to bus records in the oneline.

NOTE:Special consideration is taken when merging buses that are part of a multi-section line. If a) all buses are part of a multi-section line, b) all buses are contiguous within the multi-section line definition (all connected in series), c) include at most only one of the two multi-section line's terminal buses, and d) the resulting merge would leave at least two remaining sections of the multi-section line, then the multi-section line record will remain defined and be modified to reflect the changes incurred by merging buses and removing line sections within the record. If ANY of the above criteria are NOT met when one or more buses within a multi-section line grouping are being merged, the merge will be performed but the multi-section line record will be removed. The actual system devices will remain (in their post-merged state) but there will no longer be a multi-section line record grouping any of the devices for multi-section line reporting.

Splitting Buses

Simulator assists you in transforming one bus into two connected buses. This process is called splitting a bus. In performing the split, the user is able to decide which equipment to keep connected to the original bus and which equipment to move to the new bus. Because this activity impacts the structure of the power flow equations, bus splitting can be performed only in [Edit Mode](#).

To split a bus from the oneline diagram, right click on the bus you wish to split, and click **Split Bus** from the resulting local menu. To split a bus from the [Bus Case Information Display](#), right-click on its corresponding record and again click **Split Bus** from the resulting local menu. In either case, the [Split Bus Dialog](#) will appear.

[Multi-section lines](#) merit special consideration during bus split operations. These are the rules Simulator follows when you try to split a bus that is part of a multi-section line. If the original bus is the endpoint bus of a multi-section line and the ending line segment was transferred to the new bus, then the new bus becomes the new ending terminal of the multi-section line. If the original bus was a dummy bus of the multi-section line, and if exactly one of the branches connected to the original bus is rerouted to the new bus, then both the new bus and the original bus will be dummy buses in the reconstituted multi-section line. If neither or both of the lines connected to the original dummy bus were rerouted to the new bus, then the multi-section line definition is eliminated, since Simulator has no way to determine how the multi-section line should be redefined.

A final consideration involving bus splits is how [sequence data](#) is treated. If you have defined sequence data for fault analysis, Simulator will recalculate the sequence data for the original and new buses after the split. In this case, the zero sequence impedance for the new branch that connects the original and the new bus will be set to j0.0001.

Split Bus Dialog

The Split Bus dialog contains two sections labeled **Existing Bus** and **New Bus**. These sections enable you to designate the bus you want to split and the name and number of the new bus created by the bus split operation. If you opened the Split Bus Dialog from either the [Bus Case Information Display](#) or an oneline diagram, the Existing Bus Name and Number fields will be read-only and will identify the bus you selected. However, if you opened the Split Bus Dialog from the **Modify Case Menu** on the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab, you will first have to choose a bus to split. In this case, use the **Find...** speed button located next to the **Existing Bus** label to select the bus to split, or simply type its number in the Existing Bus Number field. Then, provide the name and number for the new bus to create using the New Bus Name and Name fields. The number you specify in the New Bus Number field must be unique; it cannot be a number that identifies another bus in the case. If you do specify an existing number, Simulator will issue an error message and require you to specify a different number. If you choose not to specify a name for the new bus, Simulator will set the name of the new bus to be the same as the bus's number.

Once you have identified the bus to split and the name and number for the new bus resulting from the split, you may then specify whether a bus tie should be inserted between the existing bus and the soon-to-be-created new bus. By checking the **Insert bus tie between existing and new buses** checkbox, you command Simulator to place a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of $0.0000 + j0.0001$ ohms. If the bus tie should be inserted as an open branch, check the **Normally open** checkbox. This option becomes available, of course, only if you elect to have Simulator automatically add the new bus tie.

After you have finished making your selections, click **OK**. Simulator will create the new bus, assign its electrical attributes to match those of the existing bus, and add the bus to the power system model. If a oneline diagram is currently active and the existing bus is represented on it, Simulator will add a symbol for the new bus to the diagram, placing it immediately to the right of the existing bus's symbol. If you elected to create a bus tie between the existing and new buses, Simulator will also add a symbol for the bus tie to the diagram.

Finally, Simulator will automatically open the [Equipment Mover Dialog](#) to help you manage the transfer of equipment from the existing bus to the new bus.

Equipment Mover

Simulator provides a convenient tool for transferring equipment between buses. Simulator allows you to move bus shunts, loads, generators, switched shunts, and transmission lines between buses. For loads, generators, and both varieties of shunts, Simulator offers you the ability to transfer all or part of the equipment from the origin bus to the destination bus.

Equipment may be transferred between buses using the Equipment Mover Dialog. The Equipment Mover Dialog can be opened in any of four ways:

- Go to the [Tools](#) ribbon tab and choose **Modify Case < Move Bus Equipment...** from the [Edit Mode](#) ribbon group.
- On the oneline display right-click on the bus from which you want to move equipment and select **Move Equipment...** from the resulting local menu.
- From a [Bus Case Information Display](#), right-click on the record corresponding to the bus from which you want to move equipment and select **Move Equipment ...** from the resulting local menu.
- As the final step of the [Split Bus](#) operation.

Regardless of the approach you take, Simulator will then open the Equipment Mover Dialog. The Equipment Mover Dialog consists of three sections. The top portion of the dialog is split in two sections that identify the bus from which equipment will be transferred (on the left) and the bus equipment will be transferred to (on the right). If the dialog was opened as the final operation of the [Split Bus](#) operation, these two buses will be hard-coded to identify the original bus and the bus resulting from the split. If the dialog was opened from the main menu, you must select both the origin and destination buses from lists that are reminiscent of [case information displays](#). If the dialog was opened using one of the other two methods, the origin bus will be set to the bus you selected, and you will then have to choose the destination bus. To select a bus to be an origin or destination bus, simply select the corresponding record from the appropriate list.

Once the origin and destination buses have been identified, you must then select the equipment to transfer from the origin to the destination bus using the case information display that occupies the bottom of the dialog. To move a particular piece of equipment, toggle the value of the **Move Object?** field to **YES**. To move just portions of loads, generators, or shunts from the origin to the destination bus, adjust the value of the **Move %** field from 100.0 to the percentage you desire.

Once you have selected the equipment you wish to transfer, click the **Move equipment** button. Simulator will adjust the power system model to reflect your equipment transfer requests. Furthermore, Simulator will provide you an opportunity to manually adjust all open oneline diagrams to reflect the equipment transfers. To facilitate this activity, Simulator will open a [Potential Misplacements Dialog](#) for each oneline that displays the origin bus. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.

If you find that you would like to reopen the [Potential Misplacements Dialog](#) after you have closed it, click the **List most recent transfers** button. This will reopen the Potential Misplacements Dialog associated with the most recent equipment transfer operation.

To conclude the equipment transfer operation, click the **Close** button.

Potential Misplacements Dialog

In the wake of an equipment transfer operation using the [Equipment Mover Dialog](#), some oneline display objects may be out of place. To ease the task of correcting these misplacements, Simulator provides the Potential Misplacements Dialog. Simulator opens a Potential Misplacements Dialog for each oneline that displays the bus that served as the origin for equipment transfer. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.

Tapping Transmission Lines

Simulator eases the process of inserting a bus at some location along an existing transmission line. This feature can be extremely useful when you want to add a new generation site to a model, for example. Rather than having to delete an existing line, place the bus, and draw two new transmission lines, Simulator simplifies the task to a one-step process.

A line can be tapped from a oneline diagram, the transmission line case information display, or by going to the [Tools](#) ribbon tab and choosing **Modify Case > Tap Transmission Line** from the [Edit Mode](#) ribbon group. From the oneline diagram, right-click on a transmission line and select **Insert Line Tap** from the popup menu. From the [case information display](#), simply right-click on the corresponding branch record and select **Tap Transmission Line** from the popup menu. Any of these methods will open the [Automatic Line Tap Dialog](#) for setting up and inserting the new bus.

Note that transmission lines may be split only from [Edit Mode](#). You cannot access this functionality from [Run Mode](#).

Automatic Line Tap Dialog

The Automatic Line Tap Dialog will allow you to define the settings to use for inserting a new bus along a transmission line. The transmission line to tap will be represented in the panel at the top of the dialog, displayed as a **Near Bus** and **Far Bus**. If the Automatic Line Tap Dialog was opened by right-clicking a line on a oneline diagram, or by right-clicking on a record in the case information display, the line will already be selected in the panel. You can change the line selection by first choosing the **Near Bus** you desire, and then selecting the **Far Bus** from the list of possible connections to the chosen **Near Bus**. Note that the percentage entered in the **Position along line** field will be in relation to the selected **Near Bus**. The inserted bus and new sections of the tapped line will adhere to the following settings:

Position along line

The field labeled **Position along line** will indicate the point where you right-clicked the mouse relative to the location of the Near Bus to the mouse-click in terms of a percentage of the total line length. In other words, if the line is 10 units long, and you clicked the right mouse at location 7 units from the Near Bus end of the line, the **Position along line** will indicate 70%. If you opened the Automatic Line Tap Dialog from the case information display or the **Tools** ribbon tab, the **Position along Line** will be set to 50%. In either case, you can adjust this setting to place the new bus more precisely. The placement of the new bus controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplied by the impedance of the original line. Note that the original charging capacitance of the line will be reassigned as determined by the selection under the **Shunt Model** option.

New Bus Number

By default, Simulator will find and set an unused bus number for you, but you can specify the number to be used for the new bus, between 1 and 2147483647. If you enter a bus number that already exists, you will be prompted to enter a different number when you click the **Tap** button.

New Bus Name

Specify a name to be assigned to the new bus. By default this field is blank, and if left blank Simulator will set the name of the bus the same as the new bus number.

New Bus Area

You can specify the area for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another value of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter an area number manually, or click **Find** to locate an area from the list of areas currently in the case. If you want a new area to be assigned to the case for this bus, simply enter an unused area number manually in the box, and Simulator will automatically set up the new area record for the case. You can then open the [Area Information Display](#) and set the name and other values for the new area.

New Bus Zone

You can specify the zone for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another zone of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter a zone number manually, or click **Find** to locate a zone from the list of zones currently in the case. If you want a new zone to be assigned to the case for this bus, simply enter an unused zone number manually in the box, and Simulator will automatically set up the new zone record for the case. You can then open the [Zone Information Display](#) and set the name and other values for the new Zone.

Shunt Model

Options determines how the charging B and G values of the transmission line are handled. By default, Simulator will use the long-line PI equivalent equations to determine appropriate values for B and G for the two new lines. Alternatively, you can assign the original charging capacitance as line shunts at the original terminal bus ends of the two new line segments in which case the charging capacitance of the two new branch elements will be set to 0.

Treat sections as a multi-section line

Check the box labeled **Treat sections as a multi-section line** to force the status of the two new line sections to be controlled in unison. Checking this box will cause a multi-section line to be created comprised of the two new line sections. See [Multi-Section Line Information](#) for details.

Click the **Tap** button to close the Automatic Line Tap Dialog and perform the line tap. If a value on the dialog is not set properly, a warning message will appear, and you will need to either change the specified value or cancel the process.

Merge Line Terminals

A transmission line can be removed from a load flow case without losing the electrical connectivity of the line's two terminal buses by merging the line terminals into a single bus. The command to merge line terminals can be performed in two ways, and can only be invoked when Simulator is in Edit mode.

First, a [MergeLineTerminals](#) script command can be invoked in Script mode or from an auxiliary file. The only parameter of the script command is a filter parameter, which must be populated with either the name of an [advanced filter](#) (with the name in quotation marks) or with the text SELECTED (no quotation marks.) If an advanced filter is given, Simulator will find all transmission lines that meet the advanced filter definition, and will individually merge the line terminals of each line one at a time. Simulator will retain the From bus information as the new merged bus information for each line, with one exception. If a line was removed by merging its terminals that was the first or last segment of a [multi-section line](#), the multi-section line terminal bus information will always be retained as the new merged bus.

The second way a line or lines can be chosen to have their line terminal buses merged is to select a set of lines in the branches table of the [model explorer](#), and to then right-click in the table and bring up the popup menu. The option to Merge Terminals is located under the Branch Records submenu.

Bus Renumbering Dialog

The Renumber Buses Display allows you to change the bus numbers for either the entire power system case, and/or for any open oneline diagrams. To show this display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group.

The bus renumbering feature is provided because it is sometimes necessary to renumber buses in the power system model, either to make room for new buses or to move buses to a different zone or area that has a different numbering scheme. It may also be necessary to renumber bus display objects on the oneline if you want to use the oneline with a case other than the one for which it was originally designed. The bus renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Bus Renumbering Dialog is used to manage the lists of current bus numbers and any desired changes to the numbering scheme. You can specify the bus numbers to change and their new values by directly typing them into the table. Alternatively, you can generate the bus list automatically by selecting one of the [Automatic Setup of Bus List Options](#) and clicking the **Setup Bus Swap List** button. The Automatic Setup options allow you to add to the list all buses in the case, all buses in the case subject to the [area/zone/owner filter](#) constraints, all buses currently displayed on the oneline, or a set of numbers from a text file.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select **Insert** from the local menu to insert a new bus number to change, or select **Delete** to delete a bus number that is currently in the list. Select **Show Dialog** to display the [Bus Information Dialog](#) corresponding to the bus number on which you right-clicked. You may clear the entire list by pressing the **Clear Bus List** button.

Once you have indicated which buses you would like to renumber in the table, select an option from **Bus Change Options** to specify where you would like to implement the changes (in both the case and the oneline, in the case only, or in the oneline only). Make the changes by pressing the **Change Bus Numbers** button. Close the dialog by pressing **Close**.

The Automatic Setup of Bus List and Bus Change options deserve further discussion. See for further details or click [NEXT](#).

Bus Renumbering: Automatic Setup of Bus List Options

This section of the [Bus Renumbering Dialog](#) is used to generate automatically a list of the buses to be renumbered. Select an option from the list and click on the **Setup Bus Swap List** button to generate the list, or **Clear Bus Swap List** to clear the list.

Load All Buses in Cases

Creates entries in the table for every bus in the system. By default the new bus number is the same as the old bus number. Of course you do not have to renumber every bus. If you would like a bus to keep its same number, either add that bus to the table, or simply have an entry with the old and new bus numbers identical.

Load Buses with Valid Area/Zone Filters

Same as the **Load All Buses In Case** option except only those buses with valid [area/zone/owner filters](#) are added. This allows you to easily modify the bus numbers in just an area or zone.

Load Only Buses on Oneline

Creates entries in the table only for those buses on the current oneline. This option is most useful when you are just changing the buses on the onelines.

Load Buses From Text File

Creates entries in the table using an external text file. The format of this file is as follows:

1 11 Specifies converting from old bus number 1 to new bus number 11

2 22 Specifies converting from old bus number 2 to new bus number 22

etc.

Freshen Current Oneline

The Freshen Current Oneline option is designed to help you quickly renumber an existing oneline to work with a new numbering scheme. You will find this method helpful if you have been using the oneline with a case and now must use it with a different case having a different set of bus numbers, but the same bus names. Freshen Current Oneline will try to match the buses on the oneline with the buses in the new case by matching bus names and kV, rather than by number (which is how Simulator usually tries to link bus display objects with bus records in the case). The best way to learn how to use Freshen Current Oneline is to consider the following example:

To update an old oneline to work with a new bus numbering scheme:

- Open the oneline and the old case with which it was used.
- Choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Load Only Buses on Oneline** and press the **Setup Bus Swap List** button.
- Change the **Swap?** field values all to *Yes*.
- Right click on the table and choose **Save List to File**. Give the file a name. For this example, we'll name the file "oldscheme.txt." This file will contain the list of buses represented on the oneline, specifying each bus's number, name, kV, and area.
- Close the old case.
- Open the new case and the oneline you wish to renumber. If any other onelines open with the case, close them. You want only the oneline you wish to renumber to be shown.

- Choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Freshen Current Online** and specify the file "oldscheme.txt" (for this example). Click the **Setup Bus Swap List** button. Simulator will match the old numbering scheme used in the online with elements in the new case by name and kV. If it finds more than one match, it will use the element's area name as a tie breaker. If it still can't reconcile the multiple matches, it will add both renumbering options to the table.
- Go through the new list and make sure that you want to swap the buses that are listed. If you do, change the **Swap?** field value for each to *Yes* (you can do this quickly for all buses by right-clicking on the **Swap?** column and choosing **Toggle All Yes**). Be sure to reconcile any duplicate bus renumbering suggestions. These are cases for which Simulator could not determine how to renumber the buses because a bus on the diagram matches more than one bus in the case by name, kV, and area.
- Click the **Change Bus Numbers** button at the bottom of the form.

Once the online has been renumbered, save it with the case by selecting **Save Case** (or **Save Case As** if you wish to give it a different name) from the [Application Button](#). See [Bus Change Options](#) for additional details.

Bus Renumbering: Bus Change Options

This option is used to specify which buses to change.

Change Both Case and Onelines (default)

Renumbers the buses in both the case and any open oneline diagrams.

Change Only Case

Only renumbers the buses in the case. The oneline bus numbers are not changed.

Change Only Onelines

Only renumbers the buses on the onelines. The case itself is not changed. You would want to select this option if you have already changed the case (or loaded a different one), but now have several onelines based on that case that also need to be changed. See [Automatic Setup of Bus List Options](#) for more details. This is the most commonly used option.

Renumber Areas/Zones/Substations Dialog

The Renumber Areas/Zones/Substations Dialog is available while in [Edit Mode](#), by going to the [Tools](#) ribbon tab, and choosing **Renumber > Renumber Areas/Zones/Substations** from the [Edit Mode](#) ribbon group. This dialog allows the renumbering of areas, zones, and substations for either the entire power system, and/or for any open oneline diagrams. This dialog is similar to the [Renumber Buses Dialog](#).

This feature is provided because it is sometimes necessary to renumber areas, zones, or substations in the power system model, either to make room for new elements or to move elements to different area, zones, or substations that have a different numbering scheme. It may also be necessary to renumber display objects on the oneline if the oneline is used with a case other than the one for which it was originally designed. The areas/zones/substations renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Renumber Areas/Zones/Substations Dialog is used to manage the list of areas, zones, and substations available for renumbering. This list is populated by selecting one of the **Automatic Setup of Swap List** options. The Automatic Setup options allow the addition of all areas, zones, or substations in the case or only those on the oneline. An option is also available to read the swap information from a file or to setup the swap list based on the current oneline. The **Types to Insert** option is available for specifying whether or not to add areas, zones, and substations to the swap list. Once the swap list option and types to insert have been selected, click the **Setup Swap List** to populate the table.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select one of the **Insert** options from the local menu to insert a new area, zone, or substation number to change, or select **Delete** to delete an element number that is currently in the list. The entire list may be cleared by clicking the **Clear Swap List** button. The **Present Number** for the element is specified in the list. Change the **New Number** field to specify the updated number of the area, zone, or substation.

The **Renumber Options** allows the specification of where the renumbering changes should be implemented. The changes can be applied to the power system case only, the oneline only, or both the case and the oneline.

Once the elements to renumber have been specified and the **New Number** fields have been set appropriately, implement the changes by pressing the **Renumber** button. The changes will be applied to the case and/or oneline as specified in the **Renumber Options**. Close the dialog by pressing **Close**.

Power Transfer Distribution Factors

The Power Transfer Distribution Factor (PTDF) display is used to calculate the incremental distribution factors associated with power transfers between two regions. These regions can be defined by areas, zones, super areas, single buses, injection groups or the system slack. These values provide a linearized approximation of how the flow on the transmission lines and interfaces change in response to transaction between the Seller and the Buyer. These values can then be visualized on the onelines using animated flows (see below for details). When considering areas, zones, or super areas as part of the transaction, the transaction for which the PTDFs are calculated is modeled by scaling the output of all generators on [AGC](#) in the source and sink areas in proportion to their relative [participation factors](#). Generators in the source area increase their output, while generators in the sink area decrease their output. When including injection groups as part of the transaction, the AGC status of any participating generators does not matter. All generators in the injection group with non-zero participation factors will be included in the transaction. When loads are included in injection groups, loads in the Seller would be decreased in order to effect a transaction and loads in the Buyer would be increased in order to effect a transaction. Of course, the PTDF calculation is a linear calculation and no changes are actually made to the output of generators or loads to study the impact of the transaction.

An important aspect to consider in calculating the PTDF is how the losses associated with the transfer are allocated. Simulator assumes that the Seller increases its injection by 100% of the transfer amount (generators increase and loads decrease), while the Buyer decreases its injection (generators decrease and loads increase) by 100% of the transaction **minus any change in system losses**. In other words, the Buyer accounts for the entire change in the system losses. Of course it is possible that a transfer may result in decreased system losses; for that case, the Buyer's change in injection will be greater than 100% of the transfer.

To Calculate the Power Transfer Distribution Factors:

- Perform an initial Power Flow Solution.
- In Run Mode, click on the Tools ribbon tab and select **Power Transfer Distribution Factors (PTDFs)** from the **Sensitivities** menu on the **Run Mode** ribbon group to open the [Power Transfer Distribution Factors Dialog](#).
- Supply the requested information on the [Power Transfer Distribution Factors Dialog](#) and click the **Calculate PTDFs** button. The distribution factors are calculated and displayed for the element set of your choice in the table at the bottom of the dialog.

The animated flows that appear on the online diagram may represent either actual flows or PTDF values. To specify that the display should show distribution factors, click the button labeled **Visualize PTDFs**. Once this button is clicked, the flow arrows on all open onelines will represent distribution factors, any [transmission line pie charts](#) will show PTDF values, and the caption of the button will change to **Visualize Actual Power Flows**. Click the button again to visualize actual power flows instead of distribution factors.

Note that when calculating PTDF values for interfaces that include contingent elements, the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). See [Line Outage Distribution Factors \(LODFs\)](#) for more information.

Power Transfer Distribution Factors Dialog

The PTDF Dialog enables you to control and to view the results of [power transfer distribution factor calculations](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Power Transfer Distribution Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

It is allowed to open two PTDF dialogs, but when doing this Simulator forces one dialog to show "**Single**" Directions, while the second dialog shows "**Multiple**" directions.

The dialog has the following options:

Linear Calculation Method

PTDFs may be calculated using either the full power flow Jacobian or only a portion of it. If you select the **Linearized AC Approximation** option, the sensitivity of the monitored element's flow will be calculated as a function of both its real and reactive power components to the voltage magnitude and angle of its terminal buses. When using the AC method, losses are included in the calculation. Simulator assumes that the change in losses is taken care of by the Buyer.

If you instead select the **Lossless DC Approximation** option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals. This method assumes that there are no losses.

The **Lossless DC with Phase Shifters Approximation** option is similar to the **Lossless DC**, except additional constraints are placed on the calculations that assume that the change in flow across active phase-shifting transformers is zero.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Directions

This option allows you to choose to define a single direction PTDF using the **Seller** and **Buyer Type** related fields, or to define multiple transfer directions between many different entities. If you choose to use multiple directions, the **Seller** and **Buyer Type** fields are replaced by the [Direction Records display](#) for viewing and defining directions.

For Single Direction: Seller Type, Buyer Type

Distribution factors can be calculated for power transfers between combinations of areas, zones, super areas, participation groups, or to a slack bus. Use the *seller type* and *buyer type* options to indicate the type of the selling and purchasing entities. These fields are only present for single direction PTDFs. When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

For Single Direction: Seller, Buyer

These drop-down boxes allow you to select the selling and buying entities. Their contents are filled when you select the seller and buyer types. These fields are only present for single direction PTDFs.

For Single Direction: Reverse Buyer/Seller

Click this button to re-calculate PTDFs for the direction that is the reverse of the direction currently shown. For example, if you have just calculated PTDFs for a transaction from area A to area B, press this button to calculate and display PTDFs for a transaction from area B to area A. This option is only present for single direction PTDFs.

Calculate PTDFs

Click on this button to update the PTDF values. The results table will reveal the latest calculations.

Automatically Recalculate

If checked, the PTDFs are automatically recalculated every time the power flow is solved.

Calculate MW-Distance

Click this button to open the **MW * Distance Calculations** form. This form allows you to calculate MW * Distance values for the transaction for which you calculated PTDFs. See [MW-Distance Calculations](#) for more information.

Increase in Losses

This is a read-only field that indicates the change in system losses caused by the transfer from the selling area to the buying area. The change is expressed as a percentage of the transfer amount. This will only be non-zero when using the **Linearized AC** calculation method.

Use Area/Zone Filters

If this box is checked, then the results table at the bottom of the dialog will include only records associated with devices located in areas or zones included in the [area/zone/owner filter](#) set.

Only Show Above %

Restricts the result set to show only those PTDFs that exceed a specified value.

Visualize Actual Power Flows, Visualize PTDFs

Select to toggle the onelines between showing the actual power flows and the PTDF flows. Selecting this button changes the Flow Visualization field for all the visible onelines. You can also change this field manually using the [Online Display Options](#) Dialog. When choosing to visualize PTDFs, the animated flow arrows will represent PTDF values, and any [transmission line pie charts](#) will also be changed to show PTDFs.

Tables of Results

The tables of results occupy the bottom of the PTDF Dialog. They are a set of [case-information displays](#) and thus share many characteristics common to all other case information displays. The tables will show results for lines/transformers, interfaces, areas, zones, generators, and phase shifters. The tables feature a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated element. You can also sort the area records by clicking on the heading of the field by which you want to sort.

Lines/Transformers

Shows the transaction distribution factors for the lines and transformers. The following fields are shown:

From Number, From Name, To Number, To Name, Circuit

Identifiers for the transmission line or transformer.

% PTDF From

Distribution factor associated with the MW flow at the "from bus" end of the line or transformer, specified as a percentage of the transaction amount.

% PTDF To

Distribution factor associated with the MW flow at the "to bus" end of the line or transformer, specified as a percentage of the transaction amount.

% Losses

Shows the percentage of the PTDF assigned as losses.

Nom KV (Max) and Nom KV (Min)

Displays the maximum and minimum nominal voltages for the line. This is useful for identifying transformers and which end of the PTDF relates to which nominal voltage.

Interfaces

Shows the transaction distribution factors for the [interface records](#). The following fields are shown:

Name

Alphanumeric identifier for the interface.

Number

Numeric identifier for the interface.

% PTDF

Distribution factor associated with the MW flow through the interface, specified as a percentage of the transaction amount. A positive value indicates the transaction would result in an increase in the flow through the interface.

Interface MW Flow

Amount of real power flowing on the interface.

Has Contingency

Signifies if an element of the interface is violating a limit.

Areas and Zones

Shows the impact the transaction has on the losses for the area or zone. The following fields are shown:

Area/Zone Number and Name

Number and name identifiers for the area or the zone

Losses %

Change in the losses in the area or zone, specified as a percentage of the transaction amount. A **positive number** indicates that the transaction would result in **increased** losses in the area or zone, while a **negative number** indicates that the transaction would result in **decreased** losses.

Gen Chg %

Total change in all of the generators in area or zone, specified as a percentage of the transaction amount. For areas, this field should show 100% in the selling area, and 100% minus the change in system losses in the buying area.

Generators

Shows the marginal participation of each generator in the transaction. The following fields are shown:

Number of Bus, Name of Bus, ID

Generator's terminal bus number and alphanumeric identifier, and the id for the generator.

Area Num of Gen, Area Name of Gen

Name and number of the generator's area.

Gen Chg %

Assumed participation of the generator in the transaction, specified as a percentage of the transaction amount. This value is directly proportional to the participation factor for the generator, provided the generator is available for AGC and is free to move in the specified direction (i.e., is not at a MW limit). The generator's participation factor and AGC status are modified on the [Generator Dialog](#), which can be displayed by right-clicking anywhere in the record's row in the table and selecting the *Show Dialog* option.

Phase Shifters

Shows the transaction distribution factors for the phase shifters. This applies when using the Lossless DC with Phase Shifters calculation method. The following fields are shown:

From Number, From Name, To Number, To Name, Circuit

Identifiers for the phase shifter.

Status

Indicates whether or not the phase shifter is in-service.

Phase (Deg)

The current phase angle of the transformer.

XF Auto

Indicates if the phase shifter is currently enabled for automatic control.

Deg per MW

This field indicates the amount of angle change, in degrees, that would be required to keep the flow across the transformer constant for a one MW transfer between the seller and the buyer.

Tap Min, Tap Max

The minimum and maximum tap positions for the phase shifter.

Directions Display

The Directions Display appears in the upper-right corner of the PTDF window when you select Multiple Directions for the PTDF Type. The Directions Display is a [case information display](#) that allows directions to be defined for performing multiple direction [Power Transfer Distribution Factors](#). The Directions Display allows you to insert, delete, and modify directions using options available from the display's [local menu](#). When directions are modified or inserted individually, the Directions Dialog will be displayed for entering the information. In addition to individually defining directions, they can also be automatically inserted using the Auto Insert Directions dialog.

Each record in the display shows the following default information:

Number, Name

A unique number and name given to the defined direction.

Source Type, Source Name

The name and type of the direction source.

Sink Type, Sink Name

The name and type of the direction sink.

Include

Determines if the direction is to be analyzed when calculating the multiple direction PTDFs.

Processed

Displays if the direction has already been processed for a multiple direction PTDF analysis.

Directions Dialog

Directions are objects that are defined and used when computing a [Power Transfer Distribution Factor](#) using multiple directions.

The **Directions Dialog** can be used to insert a new direction or to modify the information for an existing direction. This dialog can be called by choosing **Insert** or **Show Dialog** from the [Directions Display](#) local menu.

The options that can be set from this dialog include:

Name of Direction

The name for the direction. If you are entering a new direction, you can enter a new direction name. If you are modifying an existing direction, the name of the currently viewed direction will be displayed. A drop down list shows a list of currently defined direction names, and choosing one from the list will display that direction's information in the dialog. The up and down arrows next to the field also allow you to scroll through the list of defined directions.

Direction Number

If you are entering a new direction, you can put in a new direction number. If you are viewing or modifying an existing direction, the direction's number will be displayed.

Source

Select the type of the transfer direction source. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction source.

Sink

Select the type of the transfer direction sink. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction sink.

Include in list of monitored directions

Determines if the direction is to be analyzed when calculating the multiple direction PTDF's.

Auto Insert Directions

Multiple directions can be automatically inserted for PTDF studies using the **Auto Insert Directions** dialog. This option can be selected from the local menu of the [Directions Display](#) by right-clicking and choosing **Direction Records > Auto Insert Directions**.

The layout of the dialog is as follows:

Type of Direction

There are six types of directions that can be automatically defined. **Area to Slack, Zone to Slack** and **Injection Group to Slack** will define directions from areas, zones or injection groups to the slack bus of the system. **Area to Area, Zone to Zone** and **Inj. Group to Inj. Group** will define directions from areas to other areas, zones to other zones, or injection groups to other injection groups.

Delete Existing Directions

When checked, any previously defined directions will be deleted before the new directions are automatically inserted. If not checked, then automatically inserted directions will be added to the list of previously defined directions. By not deleting existing directions before automatically inserting new directions, it is possible to have more than one direction defined with the same source and sink.

Only Insert for Areas or Zones with Display Filters Set

If checked, then only Areas and Zones with their [Area/Zone/Owner Filters](#) set to **Yes** will be used when automatically inserting directions.

Starting Number, Increment By

The **Starting Number** will be the first number used when automatically numbering the automatically inserted directions. Each subsequent direction added will be numbered according to the **Starting Number** and the **Increment By** value.

Insert Directions

Insert Directions will perform the automatic insertion routine for the directions, according to the defined options.

Calculate MW-Distance

Simulator can estimate MW * Distance quantities for the system's areas and zones that result from a specified transaction. Given a transaction from a specified source to a specified sink, Simulator uses [power transfer distribution factors \(PTDFs\)](#) to estimate the change in flow for each line in the system that results from the transaction. For each line, multiplying the line's change in flow by its length then gives the MW * Distance index for that line. Simulator then sums the MW * Distance indices by area and by zone to obtain the total MW * Distance for each area and zone in response to the specified transaction.

Because the MW * Distance calculations use PTDFs, you must access the MW * Distance functionality from the PTDF Dialog. Once you have calculated PTDFs for a particular transaction by pressing the **Calculate PTDFs** button, click the **Calculate MW * Distance** button to bring up the **MW * Distance Calculations Dialog**.

The top portion of the MW * Distance Calculations Dialog is used to set the lengths of the lines in the case. Although line length is represented as a data element in the power flow case, it often is left blank. However, Simulator needs line length information if it is to calculate MW * Distance indices. Simulator offers a few options regarding the source of line length information. If you do not have access to line lengths, either from the existing case or an external text file, Simulator can estimate line lengths for you. It does this by using the Ohms/Length values you specify in the table for lines of various kV. Simply indicate the voltage levels in the first row of the table, and the corresponding ohms or reactance per length in the second column. You do not need to differentiate here between English and metric units, because the calculation is independent of the measurement system. If you want the length estimates calculated using this table to overwrite any line lengths that may already be present in the case, be sure to check the **Always Estimate Length** checkbox; otherwise, the new estimates will set the lengths only of lines whose pre-defined length isn't greater than zero. If you want the estimates to populate the lengths of lines in the model so that, when you save the model, the estimated lengths are saved as part of the model, check the **Save Estimates With Case**. (This provides a handy way to set line lengths for a case that might not have any defined.) Note that, in performing these estimates, transformers are defined as having zero length. If you do not want Simulator to estimate line lengths but instead want to use the line lengths that are currently stored in memory, check the **Do Not Use Length Estimates** box. Finally, if you want to load line lengths from a text file, click the **Load Line Lengths from File** button. This file can be either comma- or space-delimited, and each line must have the following fields in the order specified:

From_Bus_Number To_Bus_Number Circuit_ID Length

Once Simulator knows how to calculate line lengths, it can calculate MW*distance indices for each area and zone. Specify the amount of MW that will be transacted in the **Size of Transaction** textbox. You may use the arrows to increase or decrease the size of the transaction. Simulator assumes that the transaction is to occur between the source and sink groups for which you just calculated PTDFs. Press the button labeled **Calculate** to compute the indices. Two tables are populated with the results of the calculation, one for areas, and another for zones. Use the tabs to switch between the two tables. These tables are [Case Information Displays](#) and thus share common to all case information displays. Thus, you can sort the tables, add or delete columns, access the area and zone dialogs, print the tables, and save their content as HTML.

Several options can be set to customize the calculation of MW*Distance. These options are reached from the [MW*Distance Options Dialog](#).

MW-Distance Options

The calculation of MW*Distance quantities can be customized in a number of ways. These options are set from the MW*Distance Options Dialog.

Include Tie Lines Only

If this box is checked, then the only branches that contribute to the MW*Distance calculation are those that tie two areas together. Otherwise, both tie lines and lines internal to areas and zones are included in the calculation. In the latter case, tie lines are assumed to belong to the area that owns the metered end of the branch.

Internal Flows

If you choose to include both tie line flows and flows internal to areas and zones in calculating MW*Distance quantities, you have two options for how to treat internal flows. You can ignore flows resulting from the transaction that flow in the reverse direction of the existing flow on a branch by checking the **Include flow increases only** checkbox. You can also choose to treat all such counterflows as negative contributions to an area or zone's MW * Distance value by checking the **Deduct flow reductions** checkbox.

Omit Branches

To omit branches for which the PTDF corresponding to the transaction is less than a specified value, specify a nonzero percentage in this textbox.

Line Outage Distribution Factors (LODFs)

Line Outage Distribution Factors (LODFs) are a sensitivity measure of how a change in a line's status affects the flows on other lines in the system. On an energized line, the LODF calculation determines the percentage of the present line flow that will be show up on other transmission lines after the outage of the line. For example, consider an energized line, called LineX, whose present MW flow is 100 MW. If the LODFs are found to be

LODFs for LineX outage

LineX -100%

LineY + 10%

LineZ - 30%

This means that after the outage of LineX, the flow on LineX will decrease by 100 MW (of course), LineY will increase by 10 MW, and LineZ will decrease by 30 MW. The "flow on Line X" here means the flow at the *from* bus going toward the *to* bus.[Line Outage Distribution Factors \(LODFs\)](#)

Similarly, sensitivities can be calculated for the insertion of a presently open line. In this case, the LODF determines the percentage of the post-insertion line flow that will come from the other transmission line after the insertion. The "LODF" might better be named a Line Closure Distribution Factor (LCDF) in this case.

To calculate the LODFs:

- Perform an initial [Power Flow Solution](#).
- In [Run Mode](#), select **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Rune Mode** ribbon group on the [Tools](#) ribbon tab to open the [Line Outage Distribution Factors Dialog](#).
- Supply the requested information on the [Line Outage Distribution Factors Dialog](#) and click the **Calculate LODFs** button.

What else are LODFs used for?

LODFs are used extensively when modeling the linear impact of contingencies in Simulator. This is true for the calculation of [PTDFs](#) for interfaces which contain a contingent element, as well as when performing Linear [ATC analysis](#) that includes branch contingencies.

When calculating "PTDF" values for interfaces that include [contingent elements](#), the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). An OTDF is similar to PTDF, except an OTDF provides a linearized approximation of the *post-outage* change in flow on a transmission line in response to a transaction between the Seller and the Buyer. The OTDF value is a function of PTDF values and LODF values. For a single line outage, the OTDF value for line x during the outage of line y is

$$OTDF_x = PTDF_x + LODF_{x,y} * PTDF_y$$

where $PTDF_x$ and $PTDF_y$ are the PTDFs for line x and y respectively, and $LODF_{x,y}$ is the LODF for line x during the outage of line y. More complex equations are involved when studying contingencies that include multiple line outages, but the basic idea is the same.

When performing Linear [ATC analysis](#) along with calculating OTDFs, Simulator determines the linearized approximation of the post-outage flow on the line. This is similarly determined as

$$OutageFlowX = PreOutageFlowX + LODF_{x,y} * PreOutageFlowY$$

where PreOutageFlowX and PreOutageFlowY are the pre-outage flow on lines x and y .

Line Outage Distribution Factors Dialog

The LODF Dialog enables you to control and to view the results of Line Outage Distribution Factor calculations. You access this dialog by selecting **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Run Mode** ribbon group on the [Tools](#) ribbon tab in [Run Mode](#) only.

It is allowed to open two LODF dialogs, but when doing this Simulator forces one dialog to show a "**Single LODF**" calculation, while the second dialog shows the "**LODF Matrix**".

The following describes the sections of the dialog:

The dialog has the following options:

Output Option

Choose Single LODF to calculate only the sensitivities related to the status change of a single transmission line. Choose LODF matrix to calculate a matrix of sensitivities.

For Single LODF: Specify Near Bus, Far Bus

Specify the line whose status modification you would like to determine sensitivities to. The direction of the near/far chosen is important. Switching the near and far bus around will change the sign of all the LODF sensitivities.

For Single LODF: Action

Check Outage Sensitivities to determine sensitivities for the outage of a line.

Check Closure Sensitivities to determine sensitivities for the closure of a presently outaged line. The LODF might better be named a Line Closure Distribution Factor (LCDF) in this case.

For LODF Matrix: Specify Lines to Process (Outage or Closure) and Lines to Monitor

Choose which lines to process and which lines to monitor by choosing either all AC lines, only those that meet the area/zone/owner filters, only lines with Selected field set to YES, or only those lines that meet an [Advanced Filter](#). After choosing **Calculate LODFs**, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

Linear Calculation Method

Linearized AC

This calculation method is not available for LODF sensitivities.

Lossless DC

Uses the DC power flow approximation.

Lossless DC with Phase Shifters

Check this to include the impact of phase shifter controllers with the Lossless DC calculation. By checking this, it is assumed that operating phase shifters will maintain their control requirements after the line outage.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Calculate LODFs

Click this to calculate the LODFs and update the display.

Advanced LODF Calculation

Opens the [Advanced LODF Calculation](#) dialog for setting additional options for the calculation.

LODFs Tab

After calculating a LODF Matrix, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

When calculating LODFs for a single branch, This tab contains a table showing a list of the lines in the case. Since this table us another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the [local menu](#), you can print the violations, copy the violation records to the Windows clipboard for use with another application, modify the format and content of the listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

The default fields for the tab are as follows:

From Bus Number and Name

"From" bus number and name. Right- clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

To Bus Number and Name

"To" bus number and name. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses.

%LODF

The LODF value for the line.

From MW, To MW

The present MW flows on the line at the "from" bus and the "to" bus.

From CTG MW, To CTG MW

The projected MW flows after the change in line status on the line at the "from" bus and the "to" bus.

Note: LODFs are always calculated using a DC power flow technique.

Interface LODFs

This tab will only appear if calculating LODFs for a single branch and will not appear if doing the LODF matrix calculation. This tab lists LODFs for interfaces. These LODFs are calculated based on the LODFs of the individual branches that comprise the interface.

The default fields for this tab are as follows:

Number

This is the number of the given interface.

Name

This is the name of the given interface.

% LODF

This is the LODF value for the interface. This is calculated from the LODFs of the individual branches that comprise the interface. This is simply a sum of the branch LODFs based on the direction that the branch is defined in the interface and the monitored direction of the branch in the interface.

Interface MW Flow

The present MW flow on the interface.

CTG MW

The projected MW flow on the interface based on outaging the specified branch.

Advanced LODF Calculation Dialog

The Advanced LODF Calculation Dialog can be opened by clicking on the Advanced LODF Calculation button found on the [Line Outage Distribution Factors \(LODFs\) dialog](#). The advanced calculation dialog allows you to calculate the [LODFs](#) for several different contingent lines in one batch process. First, you need to use the [Contingency Analysis Tool](#) to define which contingent lines to use (all contingencies that contain a single branch outage will be used.) Also, only lines that are being monitored will have their LODFs calculated. See [Limit Monitoring Settings](#) to change which lines are monitored.

The output from the advanced LODF calculation process is saved to a file for import into another program for analysis. This dialog defines how to save the Advanced LODF results to a file for importing in another program.

Format to Save in

You can choose to save the advanced LODF results as "Monitored Branch, Contingency" pairs for PROMOD, or as a matrix in a comma-delimited text file. The comma-delimited text file is useful for loading the results into a spreadsheet program.

Only save pairs with an LODF whose absolute value is greater than

This field allows you to filter out elements whose LODF is below a certain value.

Only Include Monitored Branches whose MW flow increases

This field allows you to filter out monitored elements whose MW flow did not increase in the LODF calculation.

Maximum Columns Per Text File

This field is important when you are saving a matrix in a comma-delimited text file. Most spreadsheet programs have limits on the number of columns they can display. The default is 256, which happens to be the maximum number of columns allowed in Excel.

LODF Number Format

You can choose to have the LODF values stored in either scientific notation, or as a specified length decimal number.

File Name

You must enter a file name and location or **Browse** for a file for saving the LODF data.

Transmission Loading Relief Sensitivities

Transmission Loading Relief (TLR) Sensitivities are similar to the [Power Transfer Distribution Factors](#). Both TLR Sensitivities and PTDFs measure the sensitivity of the flow on a device to a transaction. To calculate PTDFs, you specify a source group and a sink group, and Simulator determines the percentage of a single transfer between the source and sink that flows on each of several monitored elements. For TLR sensitivities, you specify a single device, such as a transmission line, to monitor, and a group that serves either as source or as sink. Simulator then determines the sensitivity of the flow on the single monitored element to many different transactions involving the group you specified as the source or sink. To summarize, PTDFs express the sensitivity of many monitored elements to a single transaction, whereas TLR sensitivities gauge the sensitivity of a single monitored element to many different power transfers.

"TLR" stands for "Transmission Loading Relief." TLR is an industry-wide tool for managing transmission utilization to prevent overload situations that put the system at risk. For example, suppose a particular line is loaded beyond its thermal limit. Its owner will request that a TLR program be initiated, which dictates that all transactions for which 5% or more of the exchanged power flows on the overloaded element be curtailed. The TLR sensitivity tool in Simulator is useful for pinpointing those transactions that would be curtailed. Suppose we use the TLR tool to determine where area A can purchase power from while the TLR for the overloaded element is in place. We specify area A as the buyer area, identify the overloaded line, and tell Simulator to perform the calculation. Simulator will then list the sensitivity of the flow on the overloaded line to power exchanges between all other generators, areas, and buses to area A. Any transaction for which the sensitivity exceeds 5% would be curtailed; anything below 5% would be allowed to continue.

To calculate TLR Sensitivities, select **Tools > Sensitivities > TLR Sensitivities** from the main menu. This will open the [TLR Sensitivities dialog](#), which will allow you to set TLR Sensitivities options and then calculate the sensitivities.

TLR Sensitivities Dialog

The TLR Sensitivities dialog allows you to calculate [Transmission Loading Relief Sensitivities](#) for the load flow case at its solved load flow point. To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > TLR Sensitivities / Generation Shift Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

It is allowed to open two TLR Sensitivities dialogs, but when doing this Simulator forces one dialog to show the **Line/XFMR or Interface** device type, while the second dialog shows **Multiple Elements** device type.

The following describes the sections of the dialog:

Device Type

Select whether you want to calculate the sensitivities for a transmission line/transformer, for an interface, or for multiple elements. To calculate sensitivities for an interface, you must have the [interface defined](#) in the case.

When you choose an individual line/transformer or interface, you need to specify the device by selecting it from the list of devices. In this case you specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name or number. When you select the Multiple Elements option, the device selection area of the dialog changes to allow you to select a [TLR Multiple Device Type](#). The TLR results display changes accordingly in order to accommodate TLR sensitivities for multiple elements.

Line/XFMR or Interface Only: Choose device

Specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name, number, and monitored flow direction.

For multiple elements, specify the multiple device type. Choices are: selected lines/transformers, selected interfaces, overloaded lines/transformers in the base case, overloaded interfaces in the base, overloaded lines/transformers during the set of contingencies, and overloaded interfaces during the set of contingencies.

Multiple Elements Only: Select Lines/XFMRs

Opens the case information display for lines/transformers. To include a line/transformer in the TLR study, set the **Selected?** field to Yes.

Multiple Elements Only: Select Interfaces

Opens the case information display for interfaces. To include an interface in the TLR study, set the **Selected?** field to Yes.

Transactor Type

Specify if the sensitivities will be calculated for the transactor being the buyer or the seller.

Transactor Object

Specify what the transactor will be. The choices are [Area](#), [Zone](#), [Super Area](#), [Slack](#), [Injection Group](#), and [Bus](#). When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

If using the [Integrated Topology Processing add-on](#), an **Only show the primary bus for each superbuses** checkbox will be available on the Buses tab. Checking this box will limit the buses shown on that tab and the list of Bus transactors to only those that are primary buses. This provides a quick means of removing the clutter of redundant data. The primary buses and superbuses are determined from the base case topology.

TLR Sensitivities

Specify if the next set of calculated TLR sensitivities should replace the currently calculated values, or be appended to the current values. The normal option is to Clear before Calculate. When choosing to Append, then as new calculations are performed, the results will only replace the existing sensitivity value store if the new value is larger than the old value. (Note: in this situation, negative values are considered smaller than positive values).

PTDF Calculation Method

Choose the solution method to use for calculating the sensitivities similar as done on the [PTDF Dialog](#).

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

Calculate TLR Sensitivities

Click the button to calculate the sensitivities. The calculation processing through each bus and determines the sensitivity of the MW flow of selected device with respect to a change of real power at each bus. When the transactor type is buyer, then the assumption is that power is injected at each bus and absorbed at the transactor object. When the transactor type is seller, then the assumption is that power is injected at each transactor object and absorbed at each bus.

DC Model Options

Click this button to open the [DC Options](#) page. Here you can adjust the DC options you want used if you are computing the PTDFs with a Lossless DC approximation.

Include only AGCable Generators

If checked, then only generators available for automatic generation control will be included in the TLR sensitivity calculations.

Set Sensitivities At Out-Of-Service Buses Equal to Closest

Click this button to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

Line/XFMR or Interface Only: Bus, Generator, Load, Injection Group, and Area Sensitivities

The Bus Sensitivities show the values calculated at each bus. Generator sensitivities show the same information but have only a list of generators with the P Sensitivity column showing the sensitivity calculated at the terminal bus. The Load sensitivities are similar to the generator sensitivities in that they show the same information as the bus sensitivities but only show the list of loads with the P Sensitivity column giving the sensitivity calculated at the terminal bus. The Injection Group sensitivities represent averages of the bus sensitivities weighted by the participation factors of each generator, load, or switched shunt in the group. All participation factors are included in this calculation regardless of the AGC status of any included generators. The Area Sensitivities calculated represent a weighted average of the generator buses in the area with the weighting done by generator participation factor.

If using the [Integrated Topology Processing add-on](#), an **Only show the primary bus for each superbuses** checkbox will be available on the Buses tab. Checking this box will limit the buses shown on that tab and the list of Bus transactors to only those that are primary buses. This provides a quick means of removing the clutter of redundant data. The primary buses and superbuses are determined from the base case topology.

Multiple Elements Only: Multiple Bus, Generators, Loads, and Injection Group Sensitivities

The Multiple Bus Sensitivities show the values calculated at each bus for each device examined during the analysis. Multiple Generator and Load Sensitivities show the same information but have only a list of either generators or loads with the P Sensitivity column showing the sensitivity calculated at the terminal bus. The Multiple Injection Group Sensitivities represent averages of multiple bus sensitivities weighted by the participation factor of each generator, load, or switched shunt in the group. All participation factors are included in this calculation regardless of the AGC status of any included generator.

If using the [Integrated Topology Processing add-on](#), an **Only show the primary bus for each superbuses** checkbox will be available on the Buses tab. Checking this box will limit the buses shown on that tab and the list of Bus transactors to only those that are primary buses. This provides a quick means of removing the clutter of redundant data. The primary buses and superbuses are determined from the base case topology.

The results of the multiple element analysis can be accessed using script commands or SimAuto by using the variable MULTBUSTLRSENS with the appropriate data object: Bus, Gen, Load, or InjectionGroup. There will be as many variables with the MULTBUSTLRSENS name and appropriate location number as there are elements that were studied. The location numbers start at 0. The results are assigned in the order that the elements are included in the list of branches or interfaces to be studied. For example, if the list of branches includes five branches defined in the case in the following order: 1 to 2, 2 to 4, 4 to 3, 3 to 1, 2 to 3, and branches 2 to 4 and 4 to 3 are selected for inclusion in the multiple element TLR calculation, MULTBUSTLRSENS:0 will correspond to branch 2 to 4 and MULTBUSTLRSENS:1 will correspond to branch 4 to 3 for all sets of results. When selecting devices to study based on them being overloaded or contingency-overloaded, the results will appear in the order that the overloaded elements are defined with the case. To determine which elements are overloaded in the base case, use the LINELIMITPERCENT:2 field with Branch data objects and the FGPERCENT field with Interface data objects. If these fields are greater than 100 percent then the element is overloaded. The CTGVIOL variable can be used with the Branch or Interface data type to determine if an element is overloaded due to a contingency. This field will be non-zero for any element that has overloads due to contingencies.

TLR Multiple Device Type

The Multiple Device Type selector allows you to specify a group of elements for which Transmission Loading Relief (TLR) sensitivities will be calculated. The Multiple Device Type selection is accessible only when you choose the Multiple Elements option from the Device Type in the [TLR Sensitivity dialog](#).

Selected Devices

Select Lines/XFMRs

Select the Selected Devices option, and click on this button to open a dialog to choose the lines and transformers to be included in the calculation. Toggle the Selected field in the Lines and Transformer [Case Information Display](#) to Yes for the branches you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of selected Lines/XFMRs is the algebraic sum of the TLRs of each individual element. The WTLR uses as weight the current MW flow in the element.

Select Interfaces

Select the Selected Devices option, and click on this button to open a dialog to choose the interfaces to be included in the calculation. Toggle the Selected field in the interfaces [Case Information Display](#) to Yes for the interfaces you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of the selected interfaces is the sum of the TLRs of the interfaces that have a limit different from zero. The WTLR uses as weight the MW flow in the interface.

Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that are overloaded in the present case based on the [Limit Monitoring Settings and Limit Violations Dialog](#). The list of the overloaded branches and interfaces is available in the [Limit Monitoring Settings and Limit Violations](#) information display.

CTG Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that were identified as overloaded by the [Contingency Analysis](#) tool. A branch or interface is included in the calculation if it has been overloaded during at least one contingency. The list of branches and interfaces identified as overloaded can be accessed from the [View Results By Element page](#) in the [Contingency Analysis dialog](#).

The ETLR of the CTG overloaded lines and transformers is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MVA Overload of each transmission line and transformer, which is defined as the sum of the MVA overload in the line or transformer across the contingencies that caused a violation in that particular line or transformer. The Aggregate MVA Overload and a related field, the Aggregate Percent Overload of a line or transformer, are measures of the weakness of that transmission line on the grid. The Aggregated MVA Overload and the Aggregate Percent Overload are line and transformer fields that can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

The ETLR of the CTG overloaded interfaces is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MW Overload of each interface, which is defined as the sum of the MW overload of the interface across the contingencies that caused a violation in that interface. The Aggregate MW Overload and a related field, the Aggregate Percent Overload of an interface are measures of the weakness of the interface. These two interface fields can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

Effective Transmission Loading Relief (ETLR)

In the same manner as the TLR represents the MW increase in an element per MW transfer, the ETLR column in the Mult. Bus Sensitivity table represents the total MW increase in all the elements in the set per MW increase of the transaction. Let us suppose that the set contains two transmission lines: A and B and assume that for a 1 MW transfer from bus i to the transactor, the flow in line A increases in 0.5 MW and the flow in line B decreases in -0.3. Then the ETLR of bus i is +0.2 since that is the total MW increase in the elements in the set. The ETLR provides a measure of the simultaneous MW change in multiple elements, and thus the overall effect in flows on the element of the set. The ETLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

Weighted Transmission Loading Relief (WTLR)

The WTLR column in the Mult. Bus Sensitivity table weights the sensitivities based on the flow or overload flow values of the element. It is a measure of the value of a certain bus to relief transmission loading. The buses with the highest WTLR (or lowest WTLR, depending on the transactor type) are identified as the most effective buses to mitigate transmission loading considering multiple elements. The WTLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

Generation Shift Factor Sensitivities

Generation Shift Factor (GSF) Sensitivities are a specific kind of TLR calculation. GSFs always involve a transfer with the slack bus being the Buyer. Other than this, GSF and TLR calculations are *identical*. See [TLR Sensitivities](#) for more information.

Loss Sensitivities

To access the Loss Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Loss Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The **Bus Marginal Loss Sensitivities Dialog** is used to calculate and display the sensitivity of a real power loss function, P_{Losses} , to bus real and reactive power injections. Stated mathematically, the display calculates $d P_{Losses} / d P_i$ and $d P_{Losses} / d Q_i$, where P_i and Q_i are the real and reactive power injections at bus i , respectively.

Stated less formally, the display indicates how losses would change if one more MW or Mvar of power were injected at bus i . Simulator can calculate the losses for a bus relative to losses in the bus' island or area, to losses in a select group of areas, or, if the bus belongs to a super area, to losses in the bus' super area. How Simulator computes the losses is governed by the value of the **Loss FunctionType** option.

The Loss Function Type may assume one of the following six values:

Do Not Calculate Bus Loss Sensitivities

No Losses are calculated because a loss function is not specified.

Each Electrical Island

Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

Each Area

Losses are calculated with respect to the total losses for the area containing bus i . This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

Each Area or Super Area

Losses are calculated with respect to the total losses for the area containing bus i if bus i does not belong to a super area, and with respect to the total losses for the super area containing bus i if bus i does belong to a super area. When a case contains Super Areas this is more commonly done. When Simulator calculates loss sensitivities internally for use in the Economic Dispatch or the Optimal Power Flow calculation, this is the option which is used.

Areas Selected on Loss Sensitivity Form

Losses are calculated with respect to the total losses for a group of areas, specified in the Selected Areas table.

User-Specified

If you select User-Specified as the loss function type, the values last calculated using a different loss function type will become fixed. Thus you can force the loss sensitivities to remain constant when used in other features and tools of Simulator.

In steady-state power system operation, total generation must always equal total load plus losses. Therefore, the real power injection at a single bus cannot be changed arbitrarily; it must be met by a corresponding change somewhere else in the system so that the total power remains balanced. In other words, the change in power injection must somehow be absorbed. How the injection is absorbed depends on the Loss Function Type. If the Loss Function Type is **Each Island**, the injection is absorbed by the island slack. For the **Each Area** and **Selected Areas** loss functions, the injection is absorbed at the area tie-lines.

The loss sensitivities are calculated by modeling an injection of power at a bus and then assuming that this injection is absorbed by the island slack bus. The sensitivity then shows how much the losses (for the region of interest) increase when you transfer 1 MW at the injection bus to the island slack. The "region of interest" is what was chosen as Island, Each Area, or Selected Areas.

Therefore, the "absolute numbers" given by the loss sensitivity dialogs are not directly meaningful because we are always assuming that the absorbing point is the island slack bus. What is meaningful is the "difference" between sensitivity numbers.

Example:

Assume the sensitivities are calculated to be

Bus A Loss MW Sensitivity = -0.04 = Asens

Bus B Loss MW Sensitivity = -0.02 = Bsens

Bus C Loss MW Sensitivity = +0.03 = Csens

Using these we can then look at the sensitivity of generic transfers between these buses by using "superposition".

Consider the following change in injections modeling a transfer of power from Bus A to Buses B and C.

Bus A injection = +10 MW = AMW

Bus B injection = - 6 MW = BMW

Bus C injection = - 4 MW = CMW

An estimate of the change in losses can then be calculated as

$$\begin{aligned}\text{Loss Change} &= (\text{AMW})(\text{Asens}) + (\text{BMW})(\text{Bsens}) + (\text{CMW})(\text{Csens}) \\ &= (+10)(-0.04) + (-6)(-0.02) + (-4)(+0.03) \\ &= -0.4 \text{ MW}\end{aligned}$$

The Bus Marginal Loss Sensitivities Dialog houses the following controls:

Selected Areas Table

This table is used only when the **Loss Function Type** is set to *Selected Areas*; otherwise, it is ignored. Left-click on the *Include* field to include or exclude areas from the loss function.

Calculate Marginal Loss Sensitivities Button

Once the loss function type has been specified, click this button to calculate the bus marginal loss sensitivities and update the Bus Marginal Loss Sensitivities table.

Bus Marginal Loss Sensitivities Table

This table shows the bus marginal loss sensitivities for all buses with valid [Area/Zone/Owner filters](#). The Bus Marginal Loss Sensitivities Table is a type of Case Information Display and thus exhibits features and behavior similar to all other case information displays. It has a local menu from which you can choose to find out more about a particular bus. You can sort records by any of the listed fields by clicking on the column headings. The table contains the following fields:

Number, Name: Number and name of the bus.

Area Number, Area Name: Number and name of the bus' area.

Loss MW Sens: Sensitivity of the loss function to an increase in the real power injection (generated power assumed positive) at the bus.

Penalty Factors: Computed penalty factor of each bus.

MVR Sens: Sensitivity of the loss function to an increase in the reactive power injection (generated power assumed positive) at the bus.

If using the [Integrated Topology Processing add-on](#), an **Only show the primary bus for each superbuses** checkbox will be available for this table. Checking this box will limit the buses shown to only those that are primary buses. This provides a quick means of removing the clutter of redundant data. The primary buses and superbuses are determined from the base case topology.

Just Generators Marginal Loss Sensitivities Table

This table shows the bus marginal loss sensitivities for only the generator terminal buses with valid [Area/Zone/Owner filters](#). This table is otherwise identical to the displayed values and table operation as the Bus Marginal Loss Sensitivities Table described above.

Flow and Voltage Sensitivities

To access the Flows and Voltages Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Flows and Voltages Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The grids that occupy all of the tabs on this dialog lists each device of the particular type in the system, subject to the [Area/Zone/Owner Filter](#) settings. This grids are [case information displays](#), and thus shares properties and controls common to all other case information displays.

Single Meter, Multiple Transfers

The Single Meter, Multiple Transfers tab shows the effect an additional injection of real or reactive power at a bus has on various parameters. If a bus is a PV bus, instead of determining the impact that additional injection of reactive power at the bus has, the impact of changing the voltage setpoint at the bus is determined. Additionally, the sensitivities of the Flow Type will be determined due to phase shifter phase, transformer tap, and switched shunt nominal injection changes.

Device Type and Associated Flow Type

The following device types can be selected with their associated Flow Type.

Line/XFMR

MW - real power flow on the branch

Mvar - reactive power flow on the branch

MVA - apparent power flow on the branch

Interface

MW - real power flow on the interface

Mvar - reactive power flow on the interface

MVA - apparent power flow on the interface

Bus

Magnitude - voltage magnitude in per unit at the bus

Mvar - reactive power injection at the bus

Generator

Mvar - reactive power injection at the generator. When determining this sensitivity, the sensitivity of the total Mvar injection at the generator bus is determined. This is the same value that would be returned if picking Device Type of *Bus* and Flow Type of *Mvar*. This total is apportioned to specific generators at a bus based on their proportion of the total Mvar range for the bus. Only generators that are online and on AVR control are included in the calculation.

After selecting a Device Type, a particular device must be selected from the object chooser.

Whenever you make a change to any of these settings, click **Calculate Sensitivities** to update the grids with the new sensitivities.

Buses Tab

The buses tab provides sensitivities for injecting real or reactive power at a bus or changing the voltage setpoint at a bus. All injections assume that the power is absorbed at the island slack bus.

The bus fields (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Injection $dValue/dP$ --> (P Sensitivity)* gives the sensitivity of the selected Flow Type to an injection of real power at the bus selected in the row of the results
- *Sensitivity: Injection $dValue/dQ$ --> (Q Sensitivity)* gives the sensitivity of the selected Flow Type to an injection of reactive power at the bus in the row of the results. This value is only calculated for PQ buses.
- *Sensitivity: Injection $dValue/dVsetpoint$ (for PV bus) --> (V Sensitivity)* gives the sensitivity of the selected Flow Type to a change in voltage setpoint at the bus in the row of the results. This value is only calculated for PV buses.
- *Sensitivity: Injection $dAngle/dP$ (radians/MW) --> ($AngleP$ Sensitivity)* gives the sensitivity of voltage angle to an injection of real power at the bus in the row of the results. This value is only calculated when the **Device Type** is *Bus* and the **Flow Type** is *Magnitude*.
- *Sensitivity: Injection $dAngle/dQ$ (radians/Mvar) --> ($AngleQ$ Sensitivity)* gives the sensitivity of voltage angle to an injection of reactive power at the bus in the row of the results. This value is only calculated when the **Device Type** is *Bus* and the **Flow Type** is *Magnitude*.

The option labeled **Set Out-Of-Service** allows you to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

If the [Integrated Topology Processing add-on](#) is available, the **Only show the primary bus for each superbus** checkbox will be available. When checked, the buses that show up in the table will be limited to only those that are primary nodes. The determination of primary nodes is based on the base case topology. This provides an easy filter to eliminate redundant data. This option also limits the buses that appear in the chooser list when the **Device Type** is *Bus*.

Generators Tab

The generators tab provides the same sensitivity information that is available on the Buses Tab except that only generator buses are listed. This provides an easy way to filter out only generator buses.

Loads Tab

The loads tab provides the same sensitivity information that is available on the Buses Tab except that only load buses are listed. This provides an easy way to filter out only load buses.

Phase Shifters Tab

This tab provides a listing of all phase shifters in the case. The *Sensitivity: Flow/Voltage with respect to Phase --> Sensitivity/Degree* field gives the sensitivity of the **Flow Type** to a change in phase shifter phase.

LTC Transformers Tab

This tab provides a listing of all transformers in the case. The *Sensitivity: Flow/Voltage with respect to Tap --> LTC Sensitivity* field gives the sensitivity of the **Flow Type** to a change in transformer tap.

Switched Shunts Tab

This tab provides a listing of all switched shunts in the case. The *Sensitivity: Q Nominal Injection --> Q Sensitivity* field gives the sensitivity of the **Flow Type** to a change in nominal Mvar injection at a switched shunt.

Single Transfer, Multiple Meters

The Single Transfer, Multiple Meters tab shows the effect of a single transfer of real or reactive power on all the voltages in the system. To specify the transfer, choose the **Seller** and **Buyer** in the same manner done on the [Power Transfer Distribution Factors](#) dialog. Then specify whether the calculation of **Real Power (P)**, **Reactive Power (Q)**, or both are of interest.

Click **Calculate Sensitivities** to perform the calculations.

After clicking **Calculate Sensitivities**, each row of the results gives the sensitivity of voltage at the respective bus to the power transfer defined by the seller and buyer. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Transfer dV/dP (per unit/MW)* --> *VP Sensitivity* gives the sensitivity of voltage to a real power transfer
- *Sensitivity: Transfer dV/dQ (per unit/Mvar)* --> *VQ Sensitivity* gives the sensitivity of voltage to a reactive power transfer
- *Sensitivity: Transfer $dAngle/dP$ (radians/MW)* --> *AngleP Transfer Sensitivity* gives the sensitivity of voltage angle to a real power transfer
- *Sensitivity: Transfer $dAngle/dQ$ (radians/Mvar)* --> *AngleQ Transfer Sensitivity* gives the sensitivity of voltage angle to a reactive power transfer

The check box option **Set generators off AVR and continuous switched shunts to fixed at buses participating in the transfer** is also available. Selecting this option will turn off AVR control for generators and continuous switched shunts at buses containing generators participating in the transfer with a non-zero participation factor, resolve the power flow, calculate the voltage sensitivities, turn the AVR control back on for those generators and switched shunts for which it was changed, and then resolve the power flow. Stated another way, this option assumes that all generators and switched shunts at buses participating in the transfer maintain a constant reactive power output and do not maintain their voltage setpoint. With this option, affected buses become PQ buses instead of PV buses in the Jacobian. PV buses will have a zero sensitivity to injection changes because the voltage is fixed at the regulated setpoint, so in order to actually determine the impact of the injection at these buses, they need to be made into PQ buses.

If the [Integrated Topology Processing add-on](#) is available, the **Only show the primary bus for each superbust** checkbox will be available. When checked, the buses that show up in the table will be limited to only those that are primary nodes. The determination of primary nodes is based on the base case topology. This provides an easy filter to eliminate redundant data. This option also limits the buses that appear in the transactor lists when either the **Seller** or **Buyer** type is *Bus*.

Self Sensitivity

The Self Sensitivity tab shows the effect of an injection of real or reactive power at a bus on its own voltage (assuming everything is absorbed at the island slack bus). It processes through each bus calculating its own self-sensitivity only.

Each row of the results gives the sensitivity of voltage at the respective bus to a power injection at the same bus. dV/dP gives the sensitivity of voltage to real power injection and dV/dQ gives the sensitivity of voltage to reactive power injection. To speed up the calculation, sensitivities are calculated only for the buses displayed in the Bus Sensitivities grid. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Self dV/dP (per unit/MW)* --> dV/dP gives the sensitivity of voltage magnitude to an injection of real power
- *Sensitivity: Self dV/dQ (per unit/Mvar)* --> dV/dQ gives the sensitivity of voltage magnitude to an injection of reactive power
- *Sensitivity: Self $dAngle/dP$ (radians/MW)* --> $dAngle/dP$ gives the sensitivity of voltage angle to an injection of real power
- *Sensitivity: Self $dAngle/dQ$ (radians/Mvar)* --> $dAngle/dQ$ gives the sensitivity of voltage angle to an injection of reactive power

If the [Integrated Topology Processing add-on](#) is available, the **Only show the primary bus for each superbust** checkbox will be available. When checked, the buses that show up in the table will

be limited to only those that are primary nodes. The determination of primary nodes is based on the base case topology. This provides an easy filter to eliminate redundant data.

Multiple Meters, Single Control Change

The Multiple Meters, Single Control Change tab shows the impact on various quantities to a change in a control setpoint in the system.

The **Control Types** that can be selected:

- **Generator Voltage Setpoint** - sensitivity of changing the voltage setpoint at a generator in pu. Control unit is pu.
- **Transformer Tap Ratio** - sensitivity of changing the transformer's tap ratio. Control unit is the tap ratio.
- **Phase Shifter Phase Shift** - sensitivity of change the phase shifter phase angle. Control unit is degrees expect when calculating angle sensitivities and then it is radians.
- **Switched Shunt Nominal Mvar** - sensitivity of changing the nominal Mvar injection at a switched shunt. Control unit is Mvar.

Once the Control Type is selected, the specific device must be selected from the object chooser.

Click **Calculate Sensitivities** to perform the calculations.

The sensitivities that are calculated show up on the various tabs:

Buses

- *Sensitivity: $dV/dControl$ (per unit/control unit) --> $dV/dControl$* gives the sensitivity of the bus voltage in per unit to the selected Control Type
- *Sensitivity: $dAngle/dControl$ (radians/control unit) --> $dAngle/dControl$* gives the sensitivity of the bus angle in radians to the selected Control Type
- *Sensitivity: $dQ/dControl$ (Mvar/control unit) --> $dQ/dControl$* gives the sensitivity of the bus Mvar injection to the selected Control Type

If the [Integrated Topology Processing add-on](#) is available, the **Only show the primary bus for each superbuss** checkbox will be available. When checked, the buses that show up in the table will be limited to only those that are primary nodes. The determination of primary nodes is based on the base case topology. This provides an easy filter to eliminate redundant data.

Generators

- *Sensitivity: $dQ/dControl$ (Mvar/control unit) --> $dQ/dControl$* gives the sensitivity of the generator Mvar injection to the selected Control Type. When determining this sensitivity, the sensitivity of the total Mvar injection at the generator bus is determined. This total is apportioned to specific generators at a bus based on their proportion of the total Mvar range for the bus. Only generators that are online and on AVR control are included in the calculation.

Branches

- *Sensitivity: $dP/dControl$ (MW/control unit) --> $dP/dControl$* gives the sensitivity of the branch MW flow to the selected Control Type

- *Sensitivity: $dQ/dControl$ (Mvar/control unit) --> $dQ/dControl$* gives the sensitivity of the branch Mvar flow to the selected Control Type

Interfaces

- *Sensitivity: $dP/dControl$ (MW/control unit) --> $dP/dControl$* gives the sensitivity of the interface MW flow to the selected Control Type
- *Sensitivity: $dQ/dControl$ (Mvar/control unit) --> $dQ/dControl$* gives the sensitivity of the interface Mvar flow to the selected Control Type

Click **Close** to close the Flows and Voltages Dialog.

Line Loading Replicator

The Line Loading Replicator will attempt to set the loading on a transmission line or interface to a desired real power flow amount. This is accomplished by calculating distribution factors for a selected group of loads and generators to determine how the real power injection of these elements can be adjusted to produce the desired flow. If the desired flow cannot be met, the new flow amount that can be achieved and the injection changes required to meet this flow will be reported. Minimum and maximum MW limits for loads and generators will be enforced when calculating the injection changes.

To open the Line Loading Replicator, go to the [Tools](#) ribbon tab, and choose **Line Loading Replicator** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

Select Device

Device Type

Select whether to adjust the flow on a transmission line/transformer or an interface. All devices must already be defined in the case.

Device Identifier

Specify the near bus, far bus, and circuit identifier for a branch or the interface name and number for an interface. Branches will be monitored in the near bus to far bus direction and interfaces will be monitored in the direction in which they are defined to be monitored.

Present Flow

Present real power flow on the selected device.

Desired Flow

Desired real power flow on the selected device. The difference between the Desired Flow and the Present Flow (Desired Flow – Present Flow) will be the flow change for which injection changes will be calculated.

Available Injection Groups

Injection Group Identifier

Specify the injection group name and number that contains the loads and generators whose injection can be changed to reach the Desired Flow. The injection group must already be defined in the case. The injection group definition is used simply for selecting the group of elements to be used in the calculations; the participation factors defined with the participation points in the injection group are not used in the Line Loading Replicator tool.

Only Include AGCable Generation and Load

If checked, only those loads and generators in the selected injection group whose AGC status is *YES* will be included in the calculations.

Calculation Method

Select the method used to calculate the distribution factors used to determine real power injection changes.

Max and Min Load Limits for Injection Changes

Use Max and Min Load Values

If checked, the maximum and minimum load values will come from the maximum and minimum values defined with each individual load record.

Set Max and Min Load Values

Click this button to show the [Load Records Case Information Display](#). The maximum and minimum load values can be defined by displaying the **Max MW** and **Min MW** fields and setting the values as appropriate.

Use Multiplier on Present Value

If checked, the minimum and maximum value for each load will be determined by applying the **Min Multiplier** and **Max Multiplier** to the present load value.

Calculate Injection Changes

Click this button to calculate the injection changes required to meet the Desired Flow on the selected device.

Injection Changes

This is a case information display that will provide a summary of the injection changes needed, and the elements that can effect these changes, in order to meet the Flow Achieved on the selected device. The Flow Achieved will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

The results are reported in terms of injection. For generators, a positive injection means an increase in generator output and a negative injection means a decrease in generator output. For loads, the opposite is true; a positive injection means a decrease in load and a negative injection means an increase in load.

Injection Change, **Present Injection**, and **New Injection** fields will be shown by default. These Present Injection and New Injection fields are both calculated based on the present injection of the given element with the New Injection value also being dependent on the Injection Change value. The Present Injection and New Injection values will be updated any time that the present injection for an element changes in the power system. This is important to keep in mind if making any changes to the power flow case while the Line Loading Replicator dialog is open.

Injection changes are calculated so that generator and load limits are enforced. Which load limits are used is determined in the **Max and Min Load Limits for Injection Changes** option settings. The generator limits used are the minimum and maximum limits defined with each generator. These limits can be accessed from the [Generator Case Information Display](#). The load and generator limits can also be displayed in the Injection Changes table by adding the **Min MW** and **Max MW** fields to the table.

Total Injection Increase/Decrease

This value is the amount of injection increase and decrease required to meet the Flow Achieved. The net injection will be zero.

Flow Achieved

Flow amount that can be achieved by implementing the Injection Changes listed. This value will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

Implement Injection Changes

Click this button to make changes to the power flow case to match the calculated Injection Changes. Automatic generation control (AGC) and phase shifter control are disabled for the entire case when choosing to implement changes.

Implement Injection Changes and Solve Power Flow

Click this button to make changes to the power flow case to match the calculated Injection Changes and then solve the power flow. The power flow is solved with the settings that are specified in the [Power Flow Solution Options](#) specified with [Simulator Options](#). When choosing to implement changes, these options are updated so that automatic generation control (AGC) and phase shifter

control are disabled globally for the case. The Present Flow value is updated after the completion of the power flow solution.

Contingency Analysis: An Introduction

Contingency analysis is a vitally important part of any power system analysis effort. Industry planners and operators must analyze power systems covering scenarios such as the long-term effects on the transmission system of both new generation facilities and projected growth in load. Market analysts and planners must make informed decisions regarding transactions for energy trade - whether that trade is for the next hour or months down the road. PowerWorld Simulator's Contingency Analysis tools provide the ability not only to analyze a power system in its base case topology, but also to analyze the system that results from any statistically likely contingent scenario.

Industry planning and operating criteria often refer to the n-1 rule, which holds that a system must operate in a stable and secure manner following any single transmission or generation outage. In PowerWorld Simulator, the individual contingency conditions can also be tailored to consist of either a single element (such as the loss of a transmission line or transformer), or multiple elements (such as the loss of a generator, several buses and a number of branches simultaneously). See [Available Contingency Actions](#) for a complete list of possible contingency actions.

Simulator can be set to use a Full Newton solution or use a DC Load Flow method to analyze each contingency. The Full Newton approach is not as fast as a DC Load Flow, but the results tend to be significantly more accurate and allow for gauging voltage/var effects.

The [Tutorial](#) is a great place to start learning about using Simulator's Contingency Analysis Tool. We also recommend reviewing the [Terminology](#) used throughout the Contingency Analysis help files prior to continuing.

Available Contingency Actions

The current edition of Simulator can process lists of contingencies including the following:

For Transmission Lines and Transformers (Branches)

- The opening or closing of transmission lines and transformers
- [Opening all breakers](#) to isolate and open a line or transformer

For individual Generators, Loads, or Switched Shunts

- Loss or recovery of a particular generator, load, or switched shunt
- [Opening all breakers](#) to isolate and open a particular generator, load, or switched shunt

For all the Generation, Load, or Switched Shunt at a particular bus

- Movement of generation, load, or switched shunt MWs or Mvars to another terminal bus
- Changing or setting of load, switched shunt, or generator MWs or Mvars
- Change or setting of a generator or switched shunt voltage setpoint
- [Opening all breakers](#) to isolate and open a particular generator, load or switched shunt

For Buses

- Opening of all lines connected to a bus

For Interfaces

- Opening or closing of all lines or transformers in an interface
- [Opening all breakers](#) to isolate and open all lines or transformers in an interface

For Injection Groups

- Opening, closing, or changing of output of all devices in an injection group
- Setting or Changing the net MW Injection by a Percent or MW value and by proportion or by merit order
- [Opening all breakers](#) to isolate and open all devices in an injection group

Series Capacitors

- Bypassing or placing series capacitors in service
- Setting of the Series Reactance by percent or to a particular per unit value

Phase-Shifting Transformers

- Changing or setting the Phase-Shifter Regulation MW value

DC Transmission Lines

- Opening or closing DC Lines
- Changing DC Line setpoint MW or Amp values

Three-Winding Transformers

- Opening or closing all legs of three winding transformers
- [Opening all breakers](#) to isolate and open all legs of a three winding transformer

Solving the Power Flow

Contingency Block

Contingency Analysis: Terms

Contingency – Contingencies are the basis of the Contingency Analysis tool. A single contingency can contain a single contingency element (referred to as n-1 contingencies) or multiple contingency elements.

Contingency Action - A statistically likely condition that could occur during power system operation. See [Available Contingency Actions](#) for a listing of actions supported by Simulator.

Model Criteria – Criteria under which a contingency action will occur. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). Normally, no Model Criteria will be specified for a given action.

Contingency Element – Consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

Contingency Definition – A listing of the Contingency Elements assigned to a Contingency.

Global Action – A [Global Action](#) is a list of contingency elements that occur for ALL contingencies unless an Inclusion Filter is specified that allows defining with which contingencies the action will be included. The elements included in a Global Action do not have to be entered as individual elements in each contingency.

Contingency Block – A list of Contingency Actions. A [Contingency Block](#) can be applied to individual Contingencies by including the block as a Contingency Element.

Contingency Analysis Power Flow Solution Options

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options which are defined for this contingency will be used. Other options set as *Use Default* will look to the Contingency Analysis Options. Again, any options which are defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *Use Default* will be set to the same setting as the power flow solution options.

In order to specify options for a specific contingency, click on the **Define/Modify Solution Options** button on the [Contingency Definition Dialog](#). In order to specify options for all contingencies, click on the **Define/Modify Solution Options** button on the [Contingency Options: Modeling/Basics page](#). Both of these will bring up the Contingency Analysis Power Flow Solution Options dialog.

This dialog contains many options regarding the power flow solution. For options which are a numerical value, just specify a new value to use. For options which are specified by a check-box, the check-box will have three settings: *use option*, *do no use option*, and *use default*. For a more detailed explanation of each option see the [Power Flow Solution Options](#).

To set the values to be the same as the present power flow solution options, click the **Set same as for Power Flow** button. To set all options back to use default, click the **Clear All Settings** button.

Contingency Case References

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to:

- Buses
- Switched Shunts
- Limit Groups
- Loads
- Branches
- Generators
- Areas / Super Areas
- MW Transactions
- DC and Multi-Terminal DC Lines
- Power Flow Solution Options

See [Reference State Information](#) for details on the specific information stored.

The reference state is loaded into memory prior to the execution of each contingency during automatic processing of the contingency list. This ensures that all contingencies start from a common Base Case. Furthermore, the system is set back to the reference state following completion of the automatic processing. Note: The system is not restored to the reference state when the **Solve Selected Contingency** option is selected from the Contingency Record Display's local menu (see [Reference State Solution Options](#) for more information).

The reference state is always stored in Simulator after the first instance of opening the contingency analysis form. Therefore, opening the contingency analysis again may result in a prompt from the program. This prompt will ask you if you wish to set the contingency analysis reference state to the current state of the system (in case you have made changes since the last contingency analysis run), or if you wish to keep the existing contingency analysis reference state (which was set by previously opening the contingency analysis dialog.) Note that if you choose the second option, any changes you may have made to the case outside of the contingency analysis will be lost, as the reference state stored with the contingency analysis tool will reset the system state to the reference state.

Click [here](#) for information on defining the reference state.

Contingency Case References - State Information

Simulator stores the following information with the Contingency Analysis Reference State. This list should include all of the information that can be directly accessed by the user. Care should be taken with modifying any data that is not stored in the reference state as it will not be restored at the end of each contingency.

Bus State

- In or out of service
- Voltage Magnitude and angle
- Boolean expression stating whether any load exists at the bus (this is used because some of the contingency actions such as *MOVE GEN* will create a fictitious load if there is no generation at the destination bus)
- MW Marginal Cost

Switched Shunt State

- In or out of service
- Nom value MW, Mvar
- Control mode (FIXED/DISCRETE/CONTINUOUS)
- Description of Mvar blocks
- Low/high range for control regulation
- Target value for control regulation
- Discrete Control Options - Use Continuous Element, Single Largest Step, Minimum and Maximum Susceptance
- Use High Target Value and High Target Value

Limit Group State

- Rating sets for normal operation (Line, Interface...A, B, etc...)

Load State

- In or out of service
- Constant power MW and Mvar components of load
- Constant current MW and Mvar components assuming one per unit voltage
- Constant impedance MW and Mvar components assuming one per unit voltage
- MW Scale
- Mvar Scale
- AGC Status
- Min/Max Load MW

Line State

- In or out of service
- Bypassed?
- Transformer Control - Control Mode (YES/NO for transformers, Not Enabled/Enabled for Power Flow/Enabled for OPF Only for phase shifters), Regulation Minimum/Maximum, Line Drop Compensation Reactance (X), Use Line Drop Compensation
- Tap ratio

- Phase shift
- Series Reactance X (only for series capacitors because a contingency element can change this)

DC Line State

- In or out of service
- Control Mode
- Setpoint
- Setpoint voltage
- Resistance
- Compounding Resistance
- Rectifier and Inverter Tap Value
- Firing Angles (alpha and gamma)

Multi-Terminal DC Line State

- In or out of service
- Control Mode
- Bus of ac converter station controlling dc voltage
- For each converter - converter setpoint, firing angle, tap, and in or out of service

Generator State

- In or out of service
- MW Output
- Mvar Output
- Max/Min MW Output
- Participation Factor
- Max/Min Mvar Output
- Voltage Setpoint
- Regulated Bus Number
- AGC Status (YES/NO)
- AVR Status (YES/NO)
- Capability Curve
- Use Capability Curve?
- Line Drop Compensation Impedance
- Line Drop Compensation Status (YES/NO/POSTCTG)

Area State and Super Area State

- Unspecified MW Transactions
- MW Scale
- Mvar Scale
- AGC Status
- Use Area Participation Factors? (for Super Area)

MW Transaction

- MW Value
- Enabled Status

Power Flow Solution Options

The following options can be set as part of the contingency-specific power flow solution options:

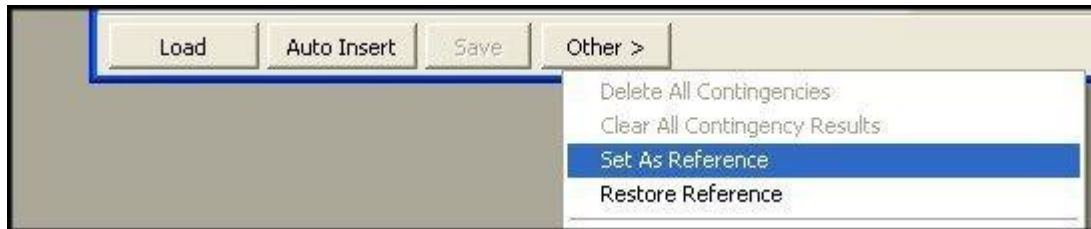
- MVA Convergence Tolerance
- Maximum Number of Iterations
- Initialize from Flat Start
- Disable Power Flow Optimal Multiplier
- Dynamically add/remove slack buses as topology is changed (Allow Multiple Islands)
- Disable Checking Gen MVAR Limits
- Check Gen MVAR Limits Immediately
- Disable Switched Shunt Control
- Disable Treating Continuous SSs as PV Buses
- Disable LTC Transformer Control
- Min. Sensitivity for LTC Control
- Disable Balancing of Parallel LTC Taps
- Disable Phase Shifter Control
- Model Phase Shifters as Discrete Controls
- Enforce Generator MW Limits
- Prevent Controller Oscillations
- Maximum Number of Voltage Control Loop Iterations
- Min. pu voltage for constant power load

The following are included as part of the reference state but are not included as contingency-specific power flow solution options:

- Min. pu voltage for constant current load
- Disable Angle Rotation Processing
- Sharing of generator vars across groups of buses during remote regulation
- Transformer Stepping Methodology
- Use Topology Processing

Contingency Case References - Defining the Reference State

The reference state is initially defined as the power system state that exists when the contingency analysis is opened for the first time for a given power flow case during a Simulator session. See [Reference State Information](#) for details on the specific information stored in the reference state.

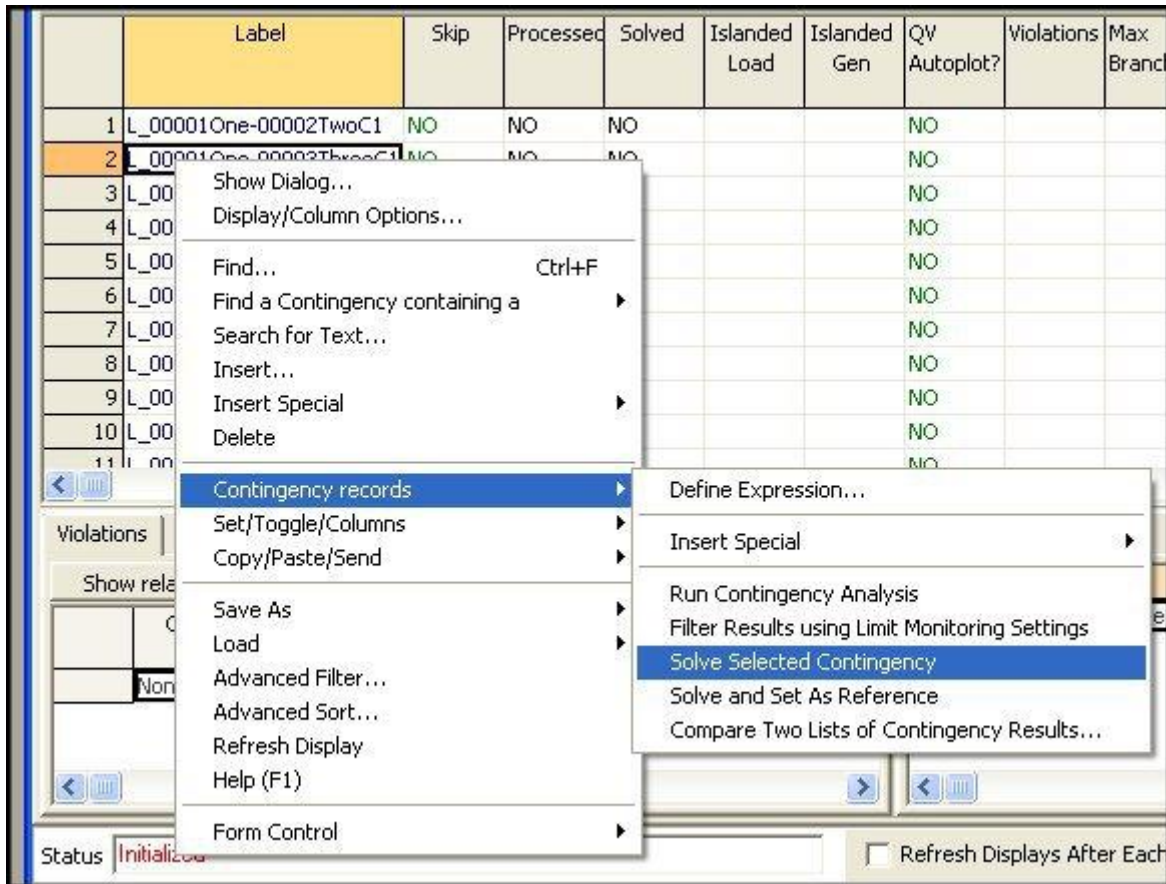


Setting the reference state

The reference state can be changed each time that the Contingency Analysis Dialog is opened after having already established a reference state. The [Setting Reference State When Contingency Analysis is Opened](#) option is available for determining how the reference state should be set when the dialog is re-opened during a Simulator session.

To change the reference while the Contingency Analysis Dialog is open, select the **Set as Reference** option from the [Other >](#) button on the [Contingencies Tab](#) of the Contingency Analysis Dialog. The reference state can also be redefined using the Solve and Set as Reference option from the Contingency Record Display's local menu (See [Reference State Solution Options](#) for more information).

Contingency Case References - Reference State Solution Options



Contingency Record Display Local Menu

When you solve contingencies one at a time, you may choose between the **Solve Selected Contingency** and **Solve and Set As Reference** options from the [Contingency Record Display's local menu](#).

Solve Selected Contingency causes Simulator to first load the reference state into memory then solve the contingency. Following the solution, *the reference state is not restored*; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

Solve and Set As Reference acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

Click [here](#) for details on the specific information stored in the reference state.

Click [here](#) for information on defining the reference state.

Contingency Analysis: Defining Contingencies

There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert a Single Element Contingency**.

Auto Insert Contingencies

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, click the **Auto Insert** button along the bottom of the [Contingency Analysis Dialog](#) or right-click on the list of contingencies table in the [Contingencies Tab](#) and select **Insert Special > Auto Insert Contingencies....** This opens the **Auto Insertion of Contingencies Dialog**.

When automatically inserting contingencies, you must specify the type, options and naming conventions you want for the new contingencies. You must also specify whether to delete or retain existing contingencies.

Auto Insertion of Contingencies Dialog

The Auto Insertion of Contingencies dialog has the following controls:

Automatically generate contingency involving a...

The options available in this box define what to add to each automatically inserted contingency element. You can choose single transmission line, transformer, transmission line or transformer, generating unit, or bus contingencies. Choosing one of these options results in each contingency containing only one element of the specific type.

You can also define contingencies containing multiple outages by checking the **Combination of...** option, and then specifying how many of each type of element (Lines, Transformers, and Generating Units) you want considered in the contingency. When you use the **Combination of...** option, Simulator will automatically determine all possible combinations for the element types specified (based on the settings in **Options** used to filter elements) and create the contingencies. Combination contingencies do not currently allow the inclusion of bus contingencies.

Restrict to Parallel or Common

These options will be enabled if choosing to do combinations where more than 1 of either a transmission line, transformer, or generator are specified.

No restrictions (All)

All combinations will be created.

Common substation

If multiple lines are to be grouped together, only combinations of lines where at least one terminal of each line is in a common substation will be included. If multiple transformers are to be grouped together, only combinations of transformers where at least one terminal of each transformer is in a common substation will be included. If multiple generators are to be grouped together, only combinations of generators that are in the same substation will be included. There is no requirement that the combinations of different types of elements are contained in the same substation. For example, combinations of lines with generators will ensure that all of the lines grouped together are in the same substations and all of the generators grouped together are in the same substation, but the groups of lines and generators grouped together can be in different substations.

Common bus

This option works the same as the **Common substation** option except that common objects are grouped such that they have a common bus instead of substation.

Parallel Branch (Common bus for gens)

If multiple lines are grouped together, only combinations of lines where both terminal buses of the lines are the same will be grouped together. If multiple transformers are grouped together, only combinations of transformers where both terminals buses of the transformers are the same will be grouped together. If multiple generators are grouped together, only combinations of generators at the same bus will be included. There is no requirement that the combinations of different types of elements be parallel. For example, combinations of lines must be parallel but they can be combined with generators that are at a different bus than either of the line terminal buses.

Action Type to Create

Two options are available for specifying how the actions for the individual contingency elements should be defined. When defining contingency actions that involving opening an element, two formats are available: OPEN or OPEN BREAKERS TO ISOLATE. The OPEN format will implement an open action by changing the status of the affected element. The OPEN BREAKERS TO ISOLATE format will identify all breakers that should be opened in order to isolate the element. This option is very useful when using a full topology model in which breakers are identified in the case. Select the **Open** option to create the OPEN contingency definitions or the **Open Breakers** option to create the OPEN BREAKERS TO ISOLATE contingency definitions.

Options

When Simulator auto-generates the contingency records for the element types specified, the options in this section will further determine which elements are included and which elements are ignored.

Delete Existing Contingencies

When checked, any previously existing contingency records will be deleted before any automatically created contingencies are inserted.

Use Area/Zone Filters

When checked, the elements included in the auto-generated contingencies will be only elements that are within areas and zones defined in the [Area/Zone/Owner Filters dialog](#). You may edit the Area/Zone/Owner filters by clicking on the **Edit Area/Zone Filters** button.

Only include ... meeting an advanced filter

When checked, the branches, generators, or buses included in the auto-generated contingencies will be only those elements meeting conditions outlined by an [advanced filter](#). To set the conditions to be used, click the respective **Define... Filter** button.

Only include elements within X buses of bus

When checked, only elements that are electrically within X number of buses from the specified bus will be included when creating the contingencies. For example, consider bus 1 is electrically connected to bus 2, which is in turn connected to bus 3. If we specify the bus to be bus 1, and choose to include only elements that are within 0 buses of bus 1, then the contingency record will include the branch between buses 1 and 2, and if desired any generators attached to bus 1. However, the branch between buses 2 and 3 and any devices attached to bus 2 and 3 will NOT be included in the contingency because bus 2 is electrically 1 bus away from bus 1.

To specify the bus used, you can find the bus by using the search engine. The search engine allows you to search by name or number. If you know the bus number, choose Sort by Number, and type the bus number in the search box. If you know the name of the bus, choose Sort by Name, and type the name of the bus in the search box. If you are not sure of the name of the bus, you can use wildcard characters to search through the list of buses until you find the desired bus.

How to name the contingencies

This section allows you to define how each automatically inserted contingency record will be named.

Identify ... using prefix

These four fields allow you to set a specific prefix for generators, lines, transformers and buses so that you can easily determine what type or types of contingencies are modeled in the auto-generated contingency records. By default, the prefixes are G for generators, L for lines, T for transformers and B for buses. However, you can change these prefixes to any character or set of characters you wish.

Identify buses by

This field allows you to specify whether each contingency is labeled using the bus numbers, bus names, or both as identifiers. Whichever type of identifier you choose here will be combined with the defined prefixes to uniquely define the individual contingency elements within each auto-generated contingency record.

Include Nominal Voltages

When this check box is checked, the nominal voltage of buses will be included in contingency labels.

Do Insert Contingencies

Press this button to generate the contingency list once all other options have been set.

Save to Aux

Press this button to save the dialog settings to an auxiliary file so that they can be recalled for later use.

Cancel

Press this button to abandon creation of contingencies and close the dialog.

Loading Contingencies from a File

Simulator can load contingency definitions from a text file. The contingencies may be specified in one of three formats:

- [Simulator Auxiliary File Format \(*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(*.con\)](#).

To load contingencies from a text file, click the **Load** button along the bottom of the [contingency analysis dialog](#). A dialog box will be provided for you to specify the file from which to load the contingency records. Specify the file type in the **Files of Type** dropdown box, and select the appropriate file. If contingency records have already been defined for the case with which you are working, you will be asked if you wish to delete the existing contingencies. Respond affirmatively to delete the existing contingencies before adding the new ones from the specified files. Otherwise, click **No**, and the contingencies loaded from the file will be appended to the already existing list.

Simulator is also capable of loading bus throw over data from [PTI PSS/E-formatted load throw over files](#).

PSS/E Contingency Format

Simulator can [read](#) and [write](#) parts of the contingency format used by Power Technology's PSS/E. The current version of Simulator supports all parts of this format, except it does not recognize PTI's Automatic Contingency Specification flags. If you need Simulator to support these keywords, contact PowerWorld Corporation to express your need. Otherwise, we recommend you make use of Simulator's [Auto-Contingency creation tool](#).

PSS/E Load Throw Over Files

Simulator can read [load throw over](#) settings used by Power Technology's PSS/E. These files typically have the extension of *.thr or *.dat. Note, however, that any modifications made to load throw over settings cannot be saved back to the PSS/E throw over file (only read access is currently available).

Saving Contingency Records to a File

Simulator can save contingency definitions to a text file. To save contingencies:

- Click the **Save** button on the [contingency analysis dialog](#). This action will save all contingency definitions and contingency options. Choosing the [Simulator Auxiliary File Format](#) provides a Contingency Settings dialog with options to save related settings such as [limit monitoring settings](#), [general power flow solution options](#), and [list display settings](#) in the same auxiliary file. Contingency Elements may also be defined in the auxiliary file using Bus Numbers, Bus Name and Nominal kV, or [Label](#).
- Select **Save As...** from the [local menu](#) of the [contingency records display](#). Several options for saving contingency information to Simulator's Auxiliary File Format are available:
 - **Auxiliary File...** This action will save the information from the columns presently shown on the contingency records display, plus the contingency elements for each contingency record.
 - **Auxiliary File (only selected records)...** This action will save the information from the columns presently shown on the contingency records display and the contingency records currently selected, plus the contingency elements for each selected contingency record.
 - **Auxiliary File (only selected records/columns)...** This action will save the information from the columns and the contingency records currently selected, plus the contingency elements for each selected contingency record.
 - **Auxiliary File (all contingency related info)...** This action is identical to that of the **Save** button on the [contingency analysis dialog](#) (see above).
 - **Auxiliary File (all contingency related info/only selected records)...** This action is identical to that of the **Save** button on the [contingency analysis dialog](#) (see above), except only definitions of the selected contingencies will be saved.
 - Options for other file formats are also available, as are typically shown on case information [local menus](#). Such other formats are generally not suitable for loading contingency information back into Simulator.
- Click the **Save to Aux** button on the Auto Insertion of Contingencies Dialog. This action will save the Auto Insertion options, but not the contingencies themselves.

The contingencies may be specified in one of three formats

- [Simulator Auxiliary File Format \(*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(*.con\)](#). This format includes an option to truncate the contingency labels to 8 or 12 characters for compatibility with versions of PSS/E that do not support longer labels. The truncated labels will be named such that each is unique. There is also an option to use bus name and bus nominal kV in identification of contingency elements.

To specify the format for the contingency file, set the **Save As Type** option accordingly.

Note that there are limitations when saving to the Simulator Version 5-7 format or the PTI PSS/E formatted files.

Limitations on the Simulator Version 5-7 format

- Does not support the actions SET or CHANGE.
- Does not support the action MOVE, except for Loads.
- Does not support any actions regarding an Interface

- Does not support any actions regarding a Bus

Limitations on the PTI PSS/E format

- Does not support any actions regarding an Interface

Contingency Global Actions

Contingency Global Actions allow you to define a list of contingency elements that occur for ALL contingencies and do not have to be entered as individual elements in each contingency. These contingency elements are defined and processed in the same manner as contingency elements defined with specific contingencies. A global action can be excluded from not being included with all contingencies by using the Inclusion Filter described below.

Contingency global actions are defined from the Contingency Global Actions display accessed via the [Contingency Definitions grouping](#) found on the [Options tab](#) of the [Contingency Analysis dialog](#). Right-click on the table to open the [Contingency Element Dialog](#), which allows the additional of contingency elements to be treated as part of the global actions.

When used with global actions, the Contingency Element Dialog will be modified to allow selecting an **Inclusion Filter** to use with a global action. The Inclusion Filter is an [advanced filter](#) or [device filter](#) that gets applied to each contingency. If the contingency meets the Inclusion Filter defined with a particular global action, that contingency will include that global action. Otherwise, a global action will be ignored for that contingency. To select an Inclusion Filter for a global action, click the **Add/Modify** button. This will open the advanced filter dialog that will allow selection of the filter. To include a global action will all contingencies, leave the Inclusion Filter blank. The Inclusion Filter is evaluated in the reference case to determine if a contingency meets the filter.

The format of the string that describes the actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of the Global Actions, check the **Do Not Use Global Action List** option found at the top of the display.

Contingency Blocks

Contingency Blocks are very similar to a [Contingency Record](#), however no results can be associated with them. Contingency Blocks consist of a list of contingency actions. The block is then given a name so that any Contingency Record can call on a Contingency Block. When a contingency block is included as part of a contingency, the Contingency Record will incorporate all the actions from the contingency block into the actions performed by the contingency.

Contingency blocks are defined from the Contingency Blocks display accessed via the [Contingency Definitions grouping](#) found on the [Options tab](#) of the [Contingency Analysis dialog](#). Right-clicking in the Contingency Blocks grid and choosing **Insert** allows you to insert a new Contingency Block. A dialog very similar to the [Contingency Definition Dialog](#) will open. Use this dialog to insert elements into the contingency block, create additional contingency blocks, or modify existing contingency blocks. Once a new contingency block has been created, actions can be added to the block by right-clicking in the Contingency Definition grid and choosing **Insert**. This will open the [Contingency Element Dialog](#) that allows the addition and modification of elements in the contingency block. When using the Contingency Element Dialog with Contingency Blocks, the option to insert an Element Type of *Contingency Block* is not available.

The format of the string that describes the contingency block actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of a Contingency Block, set the **Skip** field to *YES*. When a Contingency Block is disabled, all of the actions defined in the block will be ignored in all of the contingencies in which the block is included.

Once Contingency Blocks have been defined, a new Element Type, *Contingency Block* will appear on the [Contingency Element Dialog](#). You may then add a contingency block to a contingency by right-clicking on a contingency in the [Contingency Records](#) list display, choose show dialog (or insert if you are adding a new contingency record), click on Insert New Element, and choose Contingency Block from the list of element types.

Using a contingency block is an easy way to include a set of common actions in multiple contingency scenarios, without having to re-define the actions for each contingency.

Contingency Analysis Dialog Overview

Simulator's contingency analysis tools can be accessed only from [Run Mode](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Contingency Analysis** from the [Run Mode](#) ribbon group. This will open the Contingency Analysis dialog.

Each tab of the contingency analysis dialog covers a particular aspect of the analysis.

The [Contingencies tab](#) is used primarily to manage the contingency list and to learn basic information regarding each contingency and the violations that it causes.

The [Options tab](#) contains a panel on the left which provides access to all the options for contingency analysis. The tab enables you to dictate various parameters for the analysis that govern such things as how violations are flagged in both the Base Case and for contingency conditions and what information should be included in the contingency report. It also provides access to more complex contingency definitions and as well as Distributed Computing options.

The [Results tab](#) contains a panel on the left which provides access to look through the results of the contingency analysis. It include the ability to [View Result by Element](#) as well as by contingency. Other convenient summaries of contingency analysis results are also available as well as the ability to write results to a Text File.

For more description of how to perform a contingency analysis Simulation see [Running the Contingency Analysis](#).

Running the Contingency Analysis

To run the contingency analysis means to model and solve one or more contingencies from the case's current contingency list. Simulator's [Contingency Analysis Dialog](#) gives you several options for running the contingency analysis. You may

- Run every contingency in the contingency list (except, of course, for those you have designated to skip using the [Contingency Records Display](#)).
- Run a selected contingency to identify its limit violations and then leave the system in this post-contingency state. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state. For more information see [Contingency Case References](#)).
- Run a selected contingency to identify its limit violations and keep the resulting case as the new reference point for further contingency analysis runs.

To run the complete contingency list, do any one of the following:

- Right-click on the [Contingency Records Display](#) to bring up its [local menu](#) and select **Run Contingency Analysis**.
- Click the **Start Run** button at the bottom of the [Contingency Analysis Dialog](#).

To pause the contingency run once it has started, click either **Pause Run** at the bottom of the [Contingency Analysis Dialog](#). To resume a paused contingency run, click the **Continue** button. Finally, to terminate a contingency run, click the **Abort** button next to the **Pause/ Continue** button. The status indicator will inform you of the run's current state.

To solve a single contingency, identify its violations, and then leave the system in this post-contingency, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Contingency records > Solve Selected Contingency**. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state).

To solve a single contingency and set it as the reference (starting) case for further contingency analysis activity, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Contingency records > Solve and Set As Reference**. Simulator will model the selected contingency, flag its violations, and leave the resulting case in memory so your future work will affect the post-contingency system rather than the original pre-contingency state. See [Contingency Case References](#) for more information.

The **Refresh Displays After Each Contingency** checkbox is used to force a refresh of the counters after each contingency, which may slow down processing of the contingency set (accordingly, this box is unchecked by default to maximize solution speed).

Other Contingency Actions

By clicking the **Other >** button on the [Contingency Analysis dialog](#), you have access to several actions.

Delete All Contingencies

Click this to delete all the contingencies presently stored in memory.

Clear All Contingency Results

Click this to clear all the results of the contingencies from memory. This will not delete the contingencies.

Set As Reference

Click this to set the case presently in memory as the Reference State for contingency analysis. For more information on the reference state, see [Contingency Case References](#).

Restore Reference

Click this to set the state of the present power system case back to the contingency analysis reference state. For more information on the reference state, see [Contingency Case References](#).

Convert to Device Contingencies

This command is intended for use with full topology models, where breakers and disconnects are defined in addition to generators, loads, transmission lines, etc. This function would have no affect on a traditional planning model representation in which no breakers or disconnects are explicitly defined.

The purpose of this option is to allow the user to take a contingency set that is defined with outages of breakers and disconnects in a full topology model and convert them to outages of the traditional planning model elements, such as generators, loads, transmission lines, etc. This would be used in conjunction with the ability to save a full topology model as a consolidated model. A consolidated model reduces the full model down to a traditional planning model by examining the breaker and disconnect statuses and reducing the system down by consolidating breakers and disconnects that are in service. The resulting model is a smaller model with the traditional planning elements represented, but breakers and disconnects have been removed and nodes aggregated into bus representations. This function will also take the breaker and disconnect statuses and convert them into contingencies of the planning model devices affected by opening the original breakers or disconnects. Thus, you could create a contingency that is defined for the consolidated model and can be run on the consolidated model with the same results as if the original contingency set is run on the full topology model.

The contingency set generated depends on the statuses of the breakers and disconnects, and the contingencies created will be different for the different statuses of breakers and disconnects in the full topology model.

Combined Tables

Contingency Violation Matrices

Clicking **Process Contingency Results** will go through the existing contingency analysis tool results and build tables showing the limit violation values for all violated lines, interfaces, and buses and the contingencies for which each element is violated. Upon opening, it is possible to limit processing to branches, buses, and/or interfaces. Also, you can specify whether to display percentages or actual flows in the resulting tables.

Contingency Violation List

Contingency Violation List lists all violations from the current contingency set in a tabular format. This lists eliminates possible clutter by not listing those contingencies for which there are no violations.

Contingency Element Definitions

Contingency Violation List lists all contingency actions from the current contingency set in a tabular format.

Produce Report

To produce a detailed report of the results of the contingency analysis, click **Produce Report**. This will launch a save window that will save the information you customized on the [Report Writing](#) page. You will also be given the option of viewing the report in WordPad immediately after creating the file.

What Actually Occurred?

After running contingency analysis, click this to produce a list of actions that occurred during the selected contingency. This is especially useful for identifying those actions conditional on Model Criteria were applied.

Compare Two Lists of Contingency Results

Click this to open a dialog which allows you to specify two sets of contingency analysis results to compare. For more information on this comparison, see [Comparing Contingency Analysis Results](#).

Filter Results Using Limit Monitoring Settings

Click this to filter the contingency analysis results using the present [Limit Monitoring Settings](#). This action will go through each violation for each contingency and verify that the element that was violated is set to be monitored. If the element is not set to be monitored, then Simulator will flag the violation internally as **inactive**. You will then not be able to see these violations on any of the displays, although they will still be saved in memory until you delete these contingencies, or reprocess them. These violations will also not be saved when you choose to save the contingency results.

Because the violations are saved in memory however, you can get them back without reprocessing the contingency list. To do this, change your Limit Monitoring Settings so that those violations will be set for monitoring again. Then click this option again.

When using this option, keep in mind that Simulator cannot filter results that do not exist in memory. If the filter is to be used to filter out different percentages of overload, then the contingency analysis should be run with the lowest percentage desired. For example, if the filter option is to be used to give all of those elements that are loaded above 70% of their limit and then those that are loaded above 90% of their limit, the contingency analysis should be run with the Limit Monitoring Settings defined to monitor elements that are loaded at 70% of their limit. Also, keep in mind that the more elements are monitored the slower the processing will be.

Auto-fill Blank Contingency Element Comments

Selecting this option will fill the Comment field of the Contingency actions in the [Contingency Definition](#) table with a copy of the contingency action definition. The action description itself cannot be modified, but the comment can be modified to be more descriptive of the action being taken, for your own reference.

Sensitivity Calculations >

Simulator provides the [PTDF tool](#) for calculating the impact of a MW transfer on all the transmission lines in the system. The Simulator [ATC tool](#) further extends the linearized methods by integrating linearized contingency analysis with the PTDF calculations. The sensitivity calculations provided here are an extension of this. They allow you to ask the question, **How will each contingency-caused**

branch or interface violation be affected by a MW transfer?. Note: this calculation is not relevant for bus violations.

Calculate OTDFs using existing PTDFs

Before executing this, you must first go to the [PTDF Dialog](#) and calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Click this to calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis (see the Element description on the [Contingency Violations Display](#)).

Filter out Violations Using OTDFs

Once you have calculated the OTDFs using the existing PTDFs, you can then filter the results by selecting this option. A dialog will appear for you to enter a minimum OTDF value. All violations that have an OTDF smaller than this number will be flagged as **inactive** and will not show up in the list of violations. See the note above regarding the **Filter Results using Limit Monitoring Settings** to better understand how inactive violations are treated.

Contingency Analysis Dialog: Contingencies Tab

The Contingencies tab of the [Contingency Analysis dialog](#) provides tools for managing and simulating lists of contingencies. The top portion of the page lists the contingency records that have been defined for the case. This table is called the Contingency Records Display. The contingency records display is a type of [case information display](#) and thus shares many of the properties and controls common to all other case information displays. By right-clicking on the display, you gain access to its [local menu](#), which offers several choices:

- **Insert** allows you to insert a new contingency record
- **Insert Special** gives the following options:
 - **Quick Insert of Single Element Contingency** which allows you to quickly specify a single element contingency via the [Contingency Element Dialog](#)
 - **Auto Insert Contingencies** which opens up the [Auto Insert Contingencies](#) dialog
 - **Merge Pairs of Selected Contingencies** which generates every possible pair of contingencies based on the selected contingencies. For instance, if you select 4 contingencies (A, B, C, D), then click **Merge Pairs of Selected Contingencies**, 6 new contingencies will be generated: A+B, A+C, A+D, B+C, B+D, and C+D.
 - **Clone Contingencies** which makes copies of each selected contingency
 - **Join Active Contingencies** creates new contingencies that are the join of the current contingency list and a list read in from an auxiliary file. Contingencies with their Skip property set to YES will not be included in the join.
- **Delete** allows you to delete a particular contingency
- **Show Dialog** displays the [Contingency Definition Dialog](#) corresponding to a particular contingency
- **Contingency Records** gives the following options:
 - **Run Contingency Analysis** (see [Running the Contingency Analysis](#))
 - **Filter Results Using Limit Monitoring Settings** (see [Other Contingency Actions](#))
 - **Solve Selected Contingency** (see [Contingency Case References - Reference State Solution Options](#))
 - **Solve and Set as Reference** (see [Contingency Case References - Reference State Solution Options](#))
 - **Compare Two Lists of Contingency Results** (see [Other Contingency Actions](#))
 - **Apply Selected Contingency** which applies the actions in the contingency definition, but does not solve the contingency or calculate violations.
 - **What Actually Occurred?** (see [Other Contingency Actions](#))
- Many other options (e.g., printing, finding, and sorting) which are characteristic of [case information displays](#).
- **Save As** gives several options characteristic of case information displays as well as options unique to the contingency records display:
 - **Auxiliary File, Auxiliary File (only selected records), and Auxiliary File (only selected records/columns)** will only save contingency definitions. Any supporting data such as contingency blocks, global actions, model criteria, etc. will not be saved. To make sure that this supporting data is saved, use one of the **(all contingency related info)** options described below.
 - **Auxiliary File (all contingency related info)** first opens a file dialog for specification of the auxiliary file. Clicking Save from the file dialog opens a **Contingency Settings** dialog for specification of related settings to save in the same auxiliary file, including [Contingency Options](#), [Limit Monitoring Settings](#), [General Power Flow Solution Options](#), List Display Settings ([Case Information Customization](#)), Contingency Results, and whether or not to save unlinked elements. Definitions of all contingencies are also saved. This has the same functionality as clicking the **Save** button in the bottom panel.

- o **Auxiliary File (all contingency related/only selected records)** has the same functionality as **Auxiliary File (all contingency related info)**, except contingency definition(s) are only saved for the selected contingency record(s). You may also specify related settings to save.

You can also sort the display's contents by any field just by clicking on the field's heading. The default fields shown in the contingency records display are described at the bottom of this page.

As you scroll through the records in the contingency records display, you will notice that the contents of the tables that occupy the middle third of the contingency analysis dialog change. These tables are the [Contingency Definition Display](#) and the [Contingency Violations Display](#). These displays show the violations and definition for the contingency that is selected in the contingency records display at the top of the dialog. You may optionally hide the Contingency Definition Display by clicking on the X to the upper right of this display. To show the display again, click on the O to reopen it. You may change the relative width of the Contingency Definition and Violations display by moving your mouse over the line between the displays until your cursor changes. Then left click and drag to modify these widths.

When you first load a new contingency list into memory, the current status indicator along the bottom of the display will indicate the contingencies have been Initialized. During a contingency analysis run, the current status indicator may take on the values *Running*, *Paused*, *Aborted*, or *Finished*.

The contingency tab of the contingency analysis dialog offers several ways to run the contingency analysis. To start a run, you may click the **Start Run** button. Alternatively, you may choose **Run Contingency Analysis** from the [local menu](#) of the contingency records display. Once a contingency analysis run has started, you may pause it at any time by clicking the **Pause Run** button, after which you may resume the run by clicking **Continue**. In addition to running the full set of contingencies, you may also choose to run just a single contingency. See [Running the Contingency Analysis](#) for more details.

Several other actions related to contingency analysis are also available from the Contingencies Tab. These are accessed by clicking on the **Other Actions >** button. They are described on the [Other Contingency Actions](#) page.

You may close the Contingency Analysis Dialog at any time either by clicking **Close** or by selecting **Close** from the [local menu](#) of the Contingency Records Display or the [Contingency Violations Display](#).

By default, the contingency records display presents the following fields:

Label

The name of the contingency.

Skip

Indicates whether Simulator should skip the corresponding contingency in performing the contingency analysis. If the value of the Skip field is *Yes* for a contingency, then that contingency will not be implemented when performing the contingency analysis. This is a toggleable field, which means that you can toggle its value by double-clicking the field.

Processed

Indicates whether the contingency has been analyzed yet as part of the current contingency run.

Solved

If the contingency has not yet been processed, which means that the contingency has not yet been implemented, then the value of this field is *No*. For contingencies that have been processed, the Solved field indicates whether the power flow case that resulted from the contingency could be solved to within tolerance. If the resulting power flow case could not be solved, you should

investigate the contingency closely to determine if that contingency is indeed harmful to the stability of the system.

Post-CTG AUX

The auxiliary file specified here will be loaded at the start of this contingency, which can be used to alter the reference state for this contingency. If an auxiliary file is specified here, the contingency options [Post-Contingency Auxiliary File](#) will not be loaded for this contingency.

Double-click a cell to edit the specified post-contingency auxiliary file. A specified post-contingency auxiliary file may be removed using the [Contingency Definition Dialog](#).

Note: Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

Islanded Load

Displays the sum of the amount of load that was islanded from the rest of the system due to the contingency. This load is inaccessible from the rest of the system.

QV Autoplot?

This option must be toggled to *Yes* if you intend to run a [QV Analysis](#) and automatically plot the QV curve for one or more contingency scenarios.

Violations

Identifies the number of violations caused by the particular contingency. This number represents the total number of contingencies (branch thermal violations + bus violations + interface violations) that were caused by the contingency. Depending on how you have configured the reporting of Base Case violations (see Base Case of the Options Tab), this number may include all, some, or none of the violations that were present in the Base Case model.

Max Branch %

Indicates the percentage overload of the worst-case branch violation. If there are no branch violations, this field will be blank.

Min Volt

Indicates the lowest bus voltage resulting from the contingency. If there are no low voltage violations, this field will be blank.

Max Volt

Indicates the highest bus voltage resulting from the contingency. If there are no high voltage violations, this field will be blank.

Max Interface %

Indicates the percentage overload of the worst-case interface violation. If there are no interface violations, this field will be blank.

Contingency Definition Display

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV*, *PTI File Format*, or *Label*.

Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** found on the Model Conditions page accessed from the [Contingency Definitions grouping](#) on the Options tab of the Contingency Analysis dialog. Model Criteria can be defined from the local menu (**CTGElement records > Define Model Criteria** option) on the Contingency Definition Display, from the **Add (Model Criteria)** button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** pages in the [Contingency Definitions grouping](#) on the Options tab of the Contingency Analysis dialog.

Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under

the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

Contingency Violations Display

The Contingency Violations Display is used to list the violations that were caused by the contingency selected in the [Contingency Records Display](#).

The contingency violations display lists all of the power system elements that become violated as a result of the selected contingency. If you have selected a violation, you may click on the **Show related contingencies** button to view all contingencies that cause a violation on this power system element. Clicking this button automatically moves you to the [Results tab](#) and selects the appropriate power system element. You can also right-click on a violation in the list and choose Show Dialog from the popup menu to see the information dialog of the violated element. In addition to showing violations for branches, buses, or interfaces, results for [Custom Monitors](#) will also be displayed here. Custom Monitors are not violations, but are specific quantities that are tracked in addition to the limit monitoring.

If the contingency selected in the Contingency Records Display resulted in no violations or has not yet been processed, the Contingency Violations Display will display the words None Defined. This display is a type of [Case Information Display](#) and thus shares many characteristics and controls common to all other case information displays. You can sort the list of violations by any field simply by clicking on that field's caption.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** Selecting this will display a dialog box giving the details of the actions applied during the contingency and any actions that may have been skipped.

By default, the Contingency Violations Display contains the following fields:

Category

The type of violation that occurred. If the violation is only due to the options set in [Advanced Limit Monitoring](#), then the type will start with the word *Change*.

Valid categories include:

- Branch Amp** - branch violated due to current limit being exceeded
- Branch MVA** - branch violated due to MVA limit being exceeded
- Bus Low Volts** - bus violated due to low voltage limit being exceeded
- Bus High Volts** - bus violated due to high voltage limit being exceeded
- Bus dV/dQ** - bus violated due to dV/dQ sensitivity exceeding specified change threshold
- Bus Neg dV/dQ** - bus violated due to the dV/dQ sensitivity becoming negative
- Bus Disconnected** - bus violated due to becoming disconnected during the contingency
- Nomogram MW** - nomogram violated due to MW limit being exceeded
- Interface MW** - interface violated due to MW limit being exceeded
- Custom** - value of specific object field being monitored

Element

A character string that describes the element that suffered the violation or the element that is being monitored with a Custom Monitor. For violations, this can either be a branch, a bus, or an interface. For Custom Monitors, this can be any object type that can be monitored with the custom monitors.

When the element is a branch, this string provides you with three pieces of information:

- The branch that was violated
- The terminal of the branch which had the highest loading

- The direction of the flow on this branch

Example 1: Jamie (22) -> Amy (33) CKT 1 at Amy (33)

This means that a branch connecting Jamie(22) to Amy (33) with circuit ID 1 is violated. The violation is at the Amy(33) terminal. The -> indicates that the flow on this line is from Jamie toward Amy.

Example 2: Xena (55) <- Harley (77) CKT 1 at Harley (77)

This means that a branch connecting Xena (55) to Harley (77) with circuit ID 1 is violated. The violation is at the Harley (77) terminal. The flow on this line is from Harley toward Xena.

For Custom Monitors, the string identifies the type of object that is being monitored, the identifier of the object that is being monitored, and the variablename of the specific field that is being monitored.

Value

Indicates the value of the violating quantity. For example, if the category of the violation is *Branch Amp* and the Value field is x, then the current on the violated element is x.

For Custom Monitors this is the value of the monitored field following the contingency.

Limit

Identifies the limit value that was violated. For example, if the category of the violation is *Branch Amp* and the Limit field is y, then the limit on the current that may flow through the element is y.

For Custom Monitors this will always be zero.

Percent

The actual flow value for the element as a percentage of the limit.

For Custom Monitors this will always be blank.

Area Name Assoc.

Lists the areas with which the violated element is associated. If the element is a branch, Area Name identifies the area in which each of the branch's terminal resides. If the element is a bus, Area Name identifies the area in which the bus resides. If the element is an area-to-area interface, Area Name will identify the areas that the interface ties; otherwise, it will read *N/A*.

For Custom Monitors this will always be blank.

Nom kV Assoc.

Identifies the maximum voltage level associated with the violated element. If the violated element is a branch, then Nom kV lists the nominal voltage of its higher-voltage terminal. If the violated element is a bus, then Nom kV simply identifies the bus' nominal voltage. If the violated element is an interface that is made up strictly of branches, Nom kV lists the maximum nominal voltage of its terminals; otherwise, it will appear as *-9999.9*.

For Custom Monitors this will always be blank.

Contingency Definition Dialog

The Contingency Definition Dialog (shown below) serves as an information source for displaying the [Contingency Element](#) (or Elements) associated with individual [Contingencies](#) defined in the case. You can use the Contingency Definition Dialog to scroll through the list of elements, to view and modify their definitions, to insert new elements in a contingency or to delete a contingency. You may access this dialog by choosing either **Show Dialog** or **Insert** from the local menu of the [Contingency Records Display](#).

After making changes, click **OK** to save your changes and close the dialog. Click **Cancel** to close the dialog without saving your changes. Click **Save** to save your changes (including the addition of a new contingency) without closing the dialog (this allows you to keep working with the dialog). Click **Delete** to remove the selected contingency from the contingency list.

Contingency Definition Dialog

Contingency Label: New Contingency

Buttons: Add New, Rename, Find...

Definition: Custom, Monitoring Exceptions

Buttons: Insert New Element, Clear All

Actions	Model Criteria	Status	Comment
1 OPEN Line One 138.0 (1) TO Two 138.0 (2) CKT 1		CHECK	

Options:

- ☐ Use specific solution options
- ☐ Ignore ALL contingency specific solution options

Post-Contingency Auxiliary File

Buttons: Define Solution Options, Browse

Note: This auxiliary file is loaded at the start of each contingency. Special post-contingency settings can be entered in this auxiliary file. Note: This option will override the Advanced Contingency Option

Buttons: OK, Save, Delete, Cancel, Help

Contingency Definition Dialog

The Contingency Definition Dialog has the following controls that are available regardless of the tab that is currently selected:

Contingency Label

Identifies the name of the currently displayed contingency. Use the drop-down arrow to select a different contingency, or use the scroll buttons to navigate through the list of contingencies. When adding a new contingency, Contingency Label will show *New Contingency*. The user can change the Contingency Label by clicking **Rename**.

Add New

Click the **Add New** button to add a new [contingency](#) to the contingency list for the case. You will be prompted to enter a unique name for the new contingency. After naming the new contingency, the

name appears in the Contingency Label and you can insert new [elements](#) in the contingency definition.

Rename

Allows you to rename the selected contingency.

Find

Opens a dialog that will allow the use of [advanced search methods](#) for finding a particular contingency.

Definition Tab

Insert New Element

Click this button to add a new element to the contingency. This will open the [Contingency Element Dialog](#), used to define the Action, Model Criteria and Comment associated with the element. When you return to the Contingency Definition Dialog, the display will contain the newly inserted element.

Clear All

Removes all elements from the contingency definition. The Contingency Elements Table will then appear blank, indicating that the contingency involves no associated actions.

Definitions Display

The Contingency Definitions Display lists the [Elements](#) assigned to the selected [Contingency](#). Select **Insert** from the local menu or click on the **Insert New Element** button to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency. For more information about this display, see the [Contingency Definitions Display](#).

Define Solution Options

Click this button to open the [Contingency Solution Options Dialog](#), used to define specific power flow solutions options for use under the selected contingency.

Use Specific Solution Options

Check this box to enable the use of Contingency Specific Solution Options (see Define Solution Options above).

Ignore ALL contingency specific solution options

If checked, all contingency specific options, either defined individually for the specific contingency or defined globally in the contingency options for all contingencies, are ignored. The [solution options](#) as saved with the case will be used.

Post-Contingency Auxiliary File

The auxiliary file specified here will be loaded at the start of this contingency, which can be used to alter the reference state for this contingency. If an auxiliary file is specified here, the contingency options [Post-Contingency Auxiliary File](#) will not be loaded for this contingency. This file is applied before any [contingency-specific solution options](#) are applied. The contingency actions are then implemented after any contingency-specific solution options are applied.

Click the **Browse** button to open a file dialog for file selection. To remove the file, delete the contents of the text box.

If the file specified does not include a file path, then Simulator will look in the presently active directory, otherwise the file path specified will be used. If the post-contingency AUX file can not be found in this location, then Simulator will look to see if a file by this name exists in the directory from which the presently loading case was read. If a file is specified and still can not be

found, then when running the analysis from the contingency analysis dialog, Simulator will show a dialog requesting you to find the file or remove the option.

Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

Custom Tab

The [Custom page](#) of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the contingency. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the contingency. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

Monitoring Exceptions Tab

Monitoring exceptions can be defined for each contingency. Defining exceptions is useful when specific elements need to be excluded or included for monitoring on a contingency-by-contingency basis. Click [here](#) for more information about monitoring exceptions.

Contingency Analysis Dialog: Options Tab

The Options tab of the [Contingency Analysis dialog](#) enables you to control many parameters that govern how the contingency analysis flags violations, deals with violations that appeared in the Base Case, and documents the violations in the form of a report.

On the left of the Contingency Options tab there is a pane summarizing all the available options. This pane is split into the following 5 major groupings:

- **Modeling:** contains various options for how devices respond during a contingency which are described in the topics [Basics](#), [Generator Post-Contingency AGC](#), [Bus Load Throw Over](#), [Generator Maximum MW Response](#), [Generator Line Drop and RCC](#), and [Post-Contingency Auxiliary File](#).
- **Limit Monitoring:** contains options regarding special limit monitoring features for contingency analysis described in [Advanced Limit Monitoring](#), [Monitoring Exceptions](#), and [Custom Monitors](#).
- **Contingency Definitions** : contains listings of [contingency blocks](#) , [global actions](#), [model conditions](#), [model filters](#), [model expressions](#), and other convenient listings of contingency elements.
- **Distributed Computing** : contains special distributed computing options for contingency analysis
- **Miscellaneous**

Contingency Options: Modeling Basics

These options are all available on the [Contingency Analysis dialog](#) under the [Contingency Options tab](#) under the Modeling grouping.

Calculation Method

The calculation method defines how the power flow is solved during the contingency analysis. By default, Simulator uses a **Full Power Flow** for each contingency. For a large set of contingencies and a large case, this can take some time to complete. The Full Power Flow option can utilize either an AC solution or DC solution depending on whether or not the Power Flow Solution Options are set to [Use the DC Approximation in Power Flow](#). Alternatively, the contingency analysis can be done by calculating the linear approximation of the impact of the contingencies by choosing either the **Linearized Lossless DC** or **Linearized Lossless DC with Phase Shifters** method. These two methods will both solve a set of contingencies much faster than the full power flow. The Lossless DC methods utilize sensitivities of devices to calculate the load flow in a linear fashion. The only difference between the two lossless DC methods is that the first method treats all phase shifters as free-flowing, while the method with phase shifters will hold all in-service phase shifters at their present MW flow value.

If the Power Flow Solution Options are set to Use the DC Approximation in Power Flow, the linearized methods will not be available. Also, if the DC approximation is in use, the option with the [Limit Monitoring Settings to Treat Transmission Line Limits as Equivalent Amps](#) will be ignored. Line limits will always be reported in MVA.

Limit Monitoring Settings

Click this button to open the [Limit Monitoring Settings Dialog](#) to change the limit settings for monitored elements. See [Limit Monitoring Settings](#) for more details.

For DC methods, allow amp limits by assuming a constant voltage magnitude

If a Lossless DC calculation method has been selected from the [Modeling Tab](#), this option will become available. If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the state operating point just prior to the contingency calculations. Converting to amp limits is only enforced if the [Limit Monitoring Settings Option of Treat Transmission Line Limits as Equivalent Amps](#) is checked. When this option is checked, the contingency loading of the line will be calculated from the post-contingency current on the line and the calculated amp limit.

Retry solution using the Robust Solution Process after a contingency solution failure

Checking this option will force the contingency analysis routine to attempt a robust solution following the failure of a standard Newton-Raphson solution. The robust solution attempt does not guarantee convergence, but will attempt to slowly approach a convergence solution if possible. This process is described in more detail under [Simulation Control](#).

Note: The Robust Solution Process is not used regardless of how this option is set when solving contingencies as part of either the PV or QV analysis available with the [PVQV add-on](#).

Use specific solution options for contingencies

When checked, Simulator will use a different set of [Solution Options](#) when solving the contingencies defined in the contingency set. To define the solution options used during the contingency analysis, click the **Define Solution Options** button to open the [Contingency Solution Options Dialog](#).

Do Not Use Post Power Flow Solution Action List

Checking this option prevents any globally defined post power flow solution actions defined with the case from being evaluated and performed in the post-contingency power flow solution during contingency analysis runs.

Model reactive power for DC methods by...

When you choose to use one of the Lossless DC calculation methods, you can also specify how to handle changes in reactive power during the calculations. The lossless DC methods are based on the real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations.

The choices are:

Ignoring reactive power

Reactive power is completely ignored. This results in the MW flow only being compared to the limits of the elements in the system. It is important to recognize this fact, as branch and transformer limits are usually given in complex power (MVA) ratings, thus, ignoring reactive power results in comparing active power flow (MW) to total complex power limits (MVA.).

Assuming constant voltage magnitude

One way to include reactive power in the linearized DC results is to assume the voltage magnitudes remain constant during the linearized DC contingency analysis. Thus, the MW flows are determined from the linearized calculations, and the MVAR flows are calculated from the resulting flows and constant voltage magnitudes. Thus you will still receive an approximate complex power flow on each element (MVA), which can then be directly compared to the complex power limit of the element.

Assuming reactive power does not change

Another way to include reactive power in the linearized DC results is to assume that the reactive power magnitudes remain constant during the linearized analysis. Thus, the MW flows are determined from the linearized calculations, and the complex power flow of each element can be approximated using the calculated MW flows and the assumed constant Mvar flows from the base case. This again allows the approximate complex flow on each element to be compared to the complex power limit of the element.

Make-Up Power

When solving a contingency, the make-up power defines how the post-contingency solution accounts for the change in system losses, generation, and load. There are three options for this make-up power.

Determine make-up using:

Area Participation Factors specified below

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Calculate Participation Factors from Area Make Up Power Values** option during the contingency solution. The values specifying the **CTG Make Up Gen** for each area are used to determine the contribution of each area in an island to the amount of make-up power needed. Within each area, the individual participation factors of generators on AGC determine how much power will come from each generator.

Note that if all areas have **CTG Make Up Gen** equal to 0.0, then all make-up power will come from the island slack bus. This option generally yields faster solution times, but a concentration of make up power at the island slack bus may not accurately represent the response of the system following a contingency. The [Calculate Participation Factors from Area Make Up Power Values section of Island-Based AGC](#) describes this in further detail.

Generator Participation Factors From Entire Case Directly

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Use Participation Factors of Individual Generators** option during the contingency solution. The individual participation factors of each generator within each island are used to determine how each generator meets the make-up power needs of the island. Area and super area interchange (ACE) requirements are ignored, and all generators on AGC are dispatched to account for the changes in system losses, generation, and load. This option is usually the best approximation of post-contingent system behavior, but may yield longer solution times than island slack bus make-up power (**Area Participation Factors specified below** with **CTG Make Up Gen** set to zero for all areas).

Same as Power Flow Case

This uses the area interchange options specified in the normal power flow.

Make-up Power Tolerance

This specifies the tolerance of the [Island-Based AGC](#) used for both the **Area Participation Factors specified below** and **Generator Participation Factors From Entire Case Directly** options.

Contingency Options: Generator Post-Contingency AGC

The Generator Post-Contingency AGC display can be opened from the Modeling group of the [Contingency Analysis dialog Options page](#). This display provides a specialized generator display with specific fields used for setting the post-contingency AGC response of individual generators.

Set the **Post-CTG Prevent AGC Response** field to indicate whether or not a generator should be on AGC post-contingency. By default, generators are NOT set to prevent AGC response post-contingency. Possible options for this field are:

YES - This means that the generator will NOT respond post-contingency. AGC is disabled post-contingency regardless of the generator's AGC setting.

NO - This means that the generator will respond according to its AGC setting. If **AGC** = YES then the generator will respond post-contingency.

RESPOND - This means that the generator WILL respond post-contingency. AGC is enabled post-contingency regardless of the generator's AGC setting.

Use the **Post-CTG Part. Factor** field to specify the participation factor to be used post-contingency. A value of *same* indicates that the generator should use the same participation factor as set with the generator record **Part. Factor** field. Otherwise, set a numerical value for the post-contingency participation factor to use a factor other than the normal participation factor.

Contingency Options: Bus Load Throw Over Records

The Bus Load Throw Over list display, which is opened from the Modeling group of the [Contingency Analysis dialog Options page](#), provides you with the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system. This is referred to in Simulator as "throw over." This tool is most useful when performing contingency analysis scenarios in which buses containing loads become disconnected and you wish to analyze the impact on the system of switching the load from the disconnected bus to another bus that is still in service.

Load throw over will only attempt to move the load once, from the original bus to the load throw over bus. If the load throw over bus is already disconnected from the system, then the load will be treated as dropped during the contingency solution.

NOTE: The load throw over is only used when running the contingency analysis tool to analyze contingency effects on the system. Load throw over records are not used during manual solution of the power flow, even if you manually disconnect a bus with load and perform a load flow solution.

The Bus Load Throw Over display has the following fields:

Number, Name

The bus number and name of the load's terminal bus.

Nom kV

The nominal voltage level of the load's terminal bus.

Load Throw Over Bus Number

Enter the number of the bus you wish to have the load transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throwover Bus Name_kV** field will be automatically populated.

Load Throw Over Bus Name_kV

Enter the bus name and nominal kV (separated by a _ between the name and nominal kV) of the bus you wish the load to be transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throw Over Bus Number** will be automatically populated if the corresponding Bus Name_kV is found.

If choosing to identify objects by [labels](#), this field can also be used to display and enter the label of the load throw over bus.

Contingency Options: Generator Maximum MW Response

The Generator Maximum MW display can be opened from the Modeling group of the [Contingency Analysis dialog Options page](#). This display provides a specialized generator display with specific fields used for setting the post-contingency maximum MW response of individual generators.

Use the **CTG Max Response MW** field for entering the contingency maximum response. During the post-contingency power flow solution, a user may enter a MW amount specifying the maximum amount of generator response from a generator. By default, these values are blank. If you enter a value in this column, the generator response will be limited to this absolute MW response (response may be either an increase or a decrease). Regardless of the response settings, a generator will not respond unless its AGC field is set to YES or any [Generator Post-Contingency AGC](#) conditions are set such that the generator will be on AGC post-contingency.

The maximum MW response for a given generator will only be enforced if enforcing generator MW limits for that generator. The maximum MW response is implemented by adjusting the generator minimum and maximum limits in order to limit the response, e.g. (New Max MW) = (Present MW) + (Generator Max MW Response) and (New Min MW) = (Present MW) - (Generator Max MW Response). Therefore, the maximum MW response will only be implemented if a generator's limits are enforced.

Generator maximum MW responses will be enforced, as long as generator MW limits are also enforced, when using all contingency calculation methods including the Full Power Flow method with the power flow in either AC or DC mode and the linearized lossless DC methods.

Note: The user may also add a column called **CTG Max Response %** to the generator display. When entering data in this column, the **CTG Max Response MW** values will be set at a respective percent of the maximum MW output of the generator. If a generator's maximum MW output is less than or equal to zero, then the Maximum Response will always be set to zero.

Contingency Options: Generator Line Drop and Reactive Current Compensation

The Generator Line Drop and RCC display can be opened from the Modeling group of the [Contingency Analysis dialog Options page](#). This display provides a specialized generator display with specific fields used for setting the line drop and reactive current compensation parameters for individual generators.

Line Drop and Reactive Current Compensation represent an alternative method for performing generator voltage control. While on LDC/RCC control, the generator will vary its MVAR output in a manner that maintains the bus voltage at a fictitious bus that is a user-specified electrical impedance of **XLDC_RCC** away from the generator. This is called Line Drop Compensation when the impedance specified is positive, and Reactive Current Drop Compensation when the impedance specified is negative. If the absolute value of **XLDC_RCC** is less than or equal to $2 \times 10^{-6} \times \text{MVA Base}$, the generator will regulate its terminal bus. (For a typical case with an MVA Base of 100 MVA, this value is 0.0002.)

The impedance is specified by entering a value for **XLDC_RCC** and the setpoint voltage is the same as used when regulating a generator in the more traditional manner. A generator will perform LDC/RCC control when it meets the following conditions:

- AVR = YES
- Use LDC_RCC = YES

If any generators at a bus are set to **Use LDC_RCC**, this action will disable all traditional AVR control for generators at that bus. Other generators operating on LDC/RCC control are allowed, but no traditional AVR.

As a third special setting, the field **Use LDC_RCC** may also be set to *PostCTG* for use in contingency analysis. When a generator is set to *PostCTG* while implementing the post-contingency power flow, the generator will change the **Use LDC_RCC** value to *YES*, thereby activating this new voltage control method for the generator. After [the reference state](#) is restored in the contingency analysis however, the generator will return back to a setting of *PostCTG*.

Contingency Options: Post-Contingency Auxiliary File

Options for specifying the Post-Contingency Auxiliary File can be accessed from the Modeling group of the [Contingency Analysis dialog Options page](#). These options provide a means of specifying the post-contingency auxiliary file that can be applied to all contingencies.

The auxiliary file specified here is loaded at the start of each contingency (except for contingencies that have a [post-contingency auxiliary file specific to that contingency](#)). This file is applied before any [contingency-specific solution options](#) are applied. The contingency actions are then implemented after any contingency-specific solution options are applied. In this way, very specialized post-contingency settings can be specified. An example use of this feature could be changing the generator voltage setpoints or AVR status for the post-contingency solution or changing generator AGC status for the post-contingency.

If the file specified does not include a file path, then Simulator will look in the presently active directory, otherwise the file path specified will be used. If the post-contingency AUX file can not be found in this location, then Simulator will look to see if a file by this name exists in the directory from which the presently loading case was read. If a file is specified and still can not be found, then when running the analysis from the contingency analysis dialog, Simulator will show a dialog requesting you to find the file or remove the option.

Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

Contingency Options: Limit Monitoring

These options are all available on the [Contingency Analysis Dialog](#) under the [Contingency Options Tab](#) under the Limit Monitoring grouping. In general all the [Limit Monitoring Settings](#) specified with the case are used to determine what is eligible for monitoring. These settings can all be seen by clicking the **Limit Monitoring Settings** button. Other contingency-analysis specific limit monitoring settings are configured on this section as follows

- [Advanced Limit Monitoring](#) : the options on this page allow you to configure limit monitoring depending on how the value changes in relation to the reference case value.
- [Monitoring Exceptions](#) : the options on this page allow you to configure exceptions to the normal [Limit Monitoring Settings](#). Exceptions may cause a line to be included or excluded.
- [Custom Monitors](#) : the options on this page allow you to configure objects and associated fields to be monitored during a contingency analysis run in addition to the devices that are monitored using the [Limit Monitoring Settings](#).
- **When using Integrated Topology Processing, monitor only the primary bus for each superbus** : checking this box will reduce the repeated bus voltage limit violation that would appear for buses at the same super bus. This box is checked by default.

Contingency Options: Advanced Limit Monitoring

These options are all available on the [Contingency Analysis dialog](#) under the [Contingency Options tab](#) under the Limit Monitoring grouping.

The Advanced Limit Monitoring page allows you to shape how limit violations are detected and reported.

Many of these settings can also be specified on a Limit Group-specific basis. See the [Limit Group Dialog](#) for more information. If specified for a limit group, those settings will supersede those specified here with the contingency options.

Never report violations if...

This section controls the reporting of violations that should NEVER be reported. Minimum changes in branch flows, voltages, and interface flows which must be met before a device is reported as violating a limit can be specified. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

Increase in line/transformer flows – This is the minimum change in percentage points that the loading on a line/transformer must increase before the line/transformer gets reported as a violation. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 99% and a post-contingency loading of 100%, the line will not get reported as a violation.

Decrease in low bus voltage – This is the minimum change in a bus voltage that must occur for a bus low voltage violation to be reported. For example, if this value is set to 0.05 pu, the low voltage limit on a bus is 0.90 pu, the base case voltage at the bus is 0.91 pu, and the post-contingency voltage at the bus is 0.89 pu, this bus will not be reported as a low voltage violation.

Increase in high bus voltage – This is the minimum change in a bus voltage that must occur for bus high voltage violation to be reported. For example, if this value is set to 0.05 pu, the high voltage limit on a bus is 1.1 pu, the base case voltage at the bus is 1.09 pu, and the post-contingency voltage at the bus is 1.11 pu, this bus will not be reported as a high voltage violation.

Increase in interface flows – This is the minimum change in percentage points that the loading on an interface must increase before the interface gets reported as a violation. For example, if this value is set to 2%, interfaces are being monitored at 100%, and an interface has a base case loading of 99% and a post-contingency loading of 100%, the interface will not get reported as a violation.

Always report as a violation if...

This section allows you to specify the minimum change in flow or voltage at which point any device meeting the minimum change requirement will ALWAYS be reported, EVEN if the actual device limit (flow or voltage) is NOT violated. In other words, these options allow the reporting of large changes in flow or voltage, even if the device's actual limit is NOT itself violated. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

Increase in line/transformer flows – This is the minimum change in line/transformer flow in percentage points that the loading on a line/transformer must increase so that the line/transformer gets reported as a violation even if the loading does not exceed the element's limit. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 50% and a post-contingency loading of 63%, this line will be reported as a violation even though the post-contingency loading does not exceed the limit.

Decrease in low bus voltage – This is the minimum amount that a bus voltage must decrease for a bus low voltage violation to be reported even if the resulting voltage is higher than the bus low voltage limit. For example, if this value is set to 0.05 pu, the low voltage limit at a bus is 0.90 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 0.95 pu, this bus will be reported as a low voltage violation.

Increase in high bus voltage – This is the minimum amount that a bus voltage must increase for a bus high voltage violation to be reported even if the resulting voltage is lower than the bus high voltage limit. For example, if this value is set to 0.05 pu, the high voltage limit at a bus is 1.10 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 1.05 pu, this bus will be reported as a high voltage violation.

Increase in interface flows – This is the minimum change in interface flow in percentage points that the loading on an interface must increase so that the interface gets reported as a violation even if the loading does not exceed the interface limit. For example, if this value is set to 2%, interface limits are being monitored at 100%, and an interface has a base case loading of 75% and a post-contingency loading of 77%, this interface will be reported as a violation even though the post-contingency loading does not exceed the limit.

Caution should be used when using the **Always report...** options because this may result in a very large number of reported violations.

Report changes in bus dV/dQ sensitivity

Check this option to enable reporting changes in voltage to reactive power sensitivity for buses in the post-contingency solution. Set the sensitivity multiplier to the minimum change in sensitivity for reporting the value(s) under contingency. Any sensitivity that changes by the given multiple will be reported. You can also define an [advanced bus filter](#) using the Define Filter button, and choose either the newly defined bus filter or a previously existing bus filter for reporting dV/dQ sensitivity changes at only buses that meet the defined filter.

Also, regardless of the sensitivity multiple setting, any bus that meets the filter and has a negative dV/dQ will be reported as a violation.

Caution should be used when selecting this option. Calculating the dV/dQ sensitivities requires more computation time. Define a filter so that the dV/dQ sensitivities are only calculated for those buses for which it is important to calculate these values.

Report as a violations if a bus becomes disconnected

When checked, buses that become disconnected from the system due to a contingency will be reported as violations.

Re-reporting of base case violations

This section controls the reporting of Base Case violations. Because the concern of contingency analysis often is to identify those limitations that result directly from a particular outage or event, you may desire not to report all violations that were present in the Base Case with each contingency-specific set of violations. These options allow you to specify just how much of the Base Case violation information to report for each contingency. It gives you three options:

Do not report Base Case violations

When this option is checked, any element that was violated in the Base Case is omitted from the set of violations listed for each contingency.

List all Base Case violations for all contingencies

When this option is checked, all elements that were violated in the Base Case and are still violated post-contingency are included in the set of violations listed for each contingency.

Use these criteria

When this option is checked, only those elements that were violated in the Base Case and that meet the four criteria listed below will be listed with the contingency-specific violations. The four criteria include:

Minimum % increase in line/transformer flows: Only those branches that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum per-unit decrease in low bus voltage: Only those bus voltages that were violated in the Base Case that have decreased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum per-unit increase in high bus voltage: Only those bus voltages that were violated in the Base Case that have increased by at least this amount as a result of the contingency will be listed as contingency violations.

Minimum % increase in interface flows: Only those interfaces that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

Percentage values here are expressed in reference to a limit rather than either the base case value or post-contingency flow value. When calculating the percentage for the Base Case violations used in the comparisons, the post-contingency rating sets are used. Comparisons made using percentage changes for branches are done using percentage points and not as a percentage of the actual change. For example, if the value for the minimum percent increase in line flow is 2%, lines are monitored at 100%, and a line is loaded at 100% in the base case and, the line must be loaded to at least 102% post-contingency to be reported.

How to Monitor Voltage Changes

Voltage changes can be monitored either directly on the amount of the bus per unit voltage, or based instead on a percentage change in per unit voltage from the base case per unit voltage values.

Monitoring Exceptions

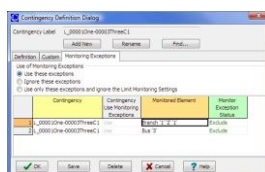
These options are all available on the [Contingency Analysis Dialog](#) under the [Contingency Options Tab](#) under the Limit Monitoring grouping.

Monitoring exceptions can be used for bus, line, or interface elements to create special rules specifying how specific elements will be monitored under a specific contingency. These rules are exceptions to the case limit monitoring options that are set via the [Limit Monitoring Settings and Limit Violations Dialog](#). Monitoring exceptions allow the inclusion or exclusion of monitoring a particular element during a specific contingency. Monitoring exceptions override the monitor status of an element as specified with the case limit monitoring settings.

Monitoring exceptions can be defined and used as given below:

Monitoring Exceptions Dialog

The Monitoring Exceptions dialog is found on the Monitoring Exceptions tab of the [Contingency Definition Dialog](#). This dialog provides a list of all exceptions that are defined for the selected contingency and provides options about how to apply the exceptions.



Use of Monitoring Exceptions

Use these exceptions - Select this option to enable use of the monitoring exceptions that are defined for this contingency. The defined exceptions will apply in addition to the case [Limit Monitoring Settings](#).

Ignore these exceptions - Select this option to disable use of the monitoring exceptions that are defined for this contingency. Only the case [Limit Monitoring Settings](#) will apply.

Use only these exceptions and ignore the Limit Monitoring Settings - Select this option to completely ignore the monitoring defined with the case [Limit Monitoring Settings](#). Only those exceptions with a **Monitor Exception Status** of *Include* will be monitored for this contingency. If the monitoring exceptions list is empty, nothing will be monitored for this contingency.

Monitoring Exceptions Table

The table lists all monitoring exceptions associated with the selected contingency. The standard fields that are listed in the table are detailed in the Monitoring Exceptions display section given below.

To insert a new monitoring exception, right-click on the table and choose **Insert** from the local menu. To edit an existing monitoring exception, right-click on the table and choose **Show Dialog** from the local menu. Either action will open the [Define Monitoring Exceptions dialog](#).

Monitoring Exceptions Display

The Monitoring Exceptions display is found on the Monitoring Exceptions subtab of the Contingency Analysis dialog [Options tab](#). The Monitoring Exceptions display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with other case information displays.

This display lists all elements that have been selected as monitoring exceptions for all contingencies. The same default fields that are listed with this table are also listed with the Monitoring Exceptions Table that is found on the Monitoring Exceptions dialog detailed above.

The default fields include:

Contingency - This is the name of the contingency for which the exception applies.

Contingency Use Monitoring Exceptions - Value of the Use of Monitoring Exceptions option that can be set for each contingency. This option can only be set from the Monitoring Exceptions dialog.

Valid entries are:

Use - Use these exceptions

Ignore - Ignore these exceptions

Only - Use only these exceptions and ignore the Limit Monitoring Settings

Monitored Element - Bus, Branch, or Interface that is an exception to the monitoring. Element is identified by object type and [key fields](#).

Monitor Exception Status - Determines how an element is monitored as an exception to the case limit monitoring settings.

Valid entries are:

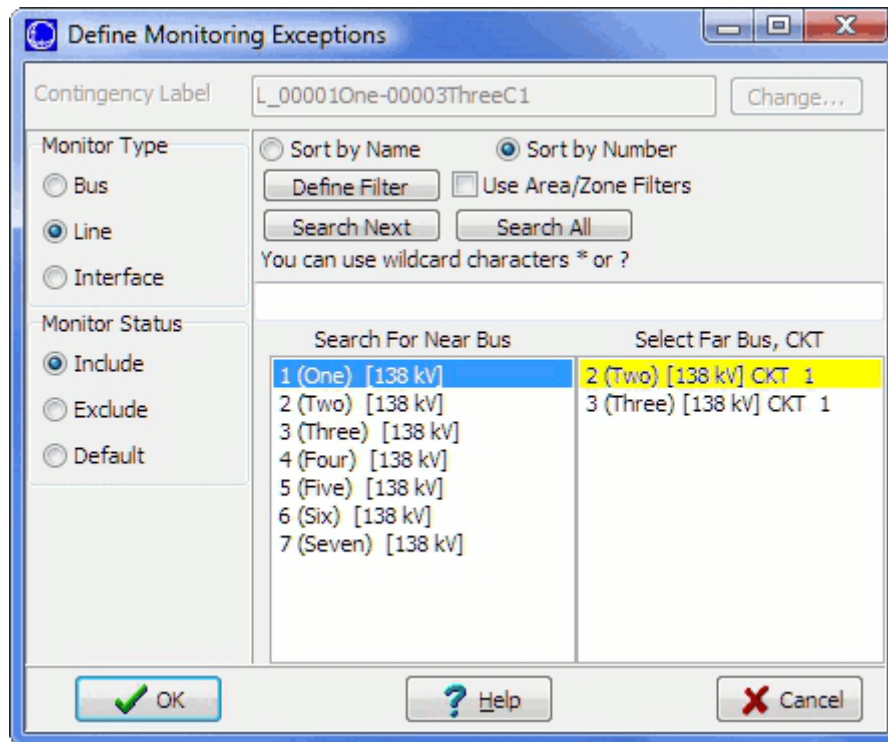
Include - The selected object will be included for monitoring with this contingency regardless of its status in the case limit monitoring settings.

Exclude - The selected object will be excluded in the monitoring with this contingency regardless of its status in the case limit monitoring settings.

Default - The selected object will obey its monitoring status in the case limit monitoring settings. Use this option when a particular monitoring exception should be ignored for a particular case but the object should remain in the list of monitoring exceptions.

Define Monitoring Exceptions Dialog

This dialog is used to define exceptions for monitoring elements during specific contingencies. This dialog will open when choosing to insert a new exception or changing an exception from either the [Monitoring Exceptions dialog](#) or the [Monitoring Exceptions display](#).



Contingency Label

This gives the contingency for which the monitoring exception applies. Click the **Change** button to search through a list of all contingencies and select a different contingency. If inserting or changing monitoring exceptions from the [Monitoring Exceptions dialog](#) that is shown with a given contingency, the contingency cannot be changed and the Change button will not be enabled.

Monitor Type

Three types of elements can be used for monitoring exceptions: bus, line, and interface. When changing the Monitor Type, the list of objects in the Object Selector will be updated accordingly.

Monitor Status

Include - The selected object will be included for monitoring with this contingency regardless of its status in the case limit monitoring settings.

Exclude - The selected object will be excluded in the monitoring with this contingency regardless of its status in the case limit monitoring settings.

Default - The selected object will obey its monitoring status in the case limit monitoring settings. Use this option when a particular monitoring exception should be ignored for a particular case but the object should remain in the list of monitoring exceptions.

Object Selector

Use this to select the object for the monitoring exception. Lists all objects in the case of the type selected under Monitor Type. Use the [filtering and search methods](#) to better manage the list in making a selection.

OK, Cancel

Click OK to accept the monitoring exception changes for the selected contingency and close the dialog. Click Cancel to close the dialog without accepting the changes.

Custom Monitors

Custom Monitors can be defined, modified, and viewed from the [Contingency Analysis dialog](#) under the [Contingency Options tab](#) under the [Limit Monitoring](#) page. Custom Monitors are also available in the [Model Explorer](#) in the Contingency Analysis folder under Custom Monitors.

Custom Monitors can be used to monitor specific objects and fields in addition to the devices that are monitored using the [Limit Monitoring Settings](#). Custom monitors do not represent violations, but rather, specific object fields that are of interest during the contingency analysis. Results for custom monitors will appear in the [Contingency Violations Display](#) and [Contingency Results tab](#) on the [View Results By Element page](#).

The Custom Monitors table is a type of [case information display](#) and thus shares many of the properties and controls common to all other case information displays. Existing custom monitors can be edited directly in the table or by choosing Show Dialog either from the [local menu](#) obtained by right-clicking or choosing this option from the [Case Information Toolbar](#). To create a new Custom Monitor, choose Insert from the local menu or Case Information Toolbar.

The dialog for creating and modifying a custom monitor has the following options:

Element Type

Type of element to be monitored. One must be selected. Available elements types include: Area, Branch, Bus, DC Transmission Line, Generator, Injection Group, Interface, Limit Set, Load, Model Expression, Multi-Terminal DC Record, Nomogram, Owner, Substation, Super Area, Switched Shunt, Transformer, and Zone.

Custom Monitor Name

Name of the custom monitor. Each custom monitor must have a unique name. A default name will be created based on existing custom monitors. Click the **Rename** button to open a dialog that will allow you to change the name.

Enabled

Check this box for the specified object(s) and field to be monitored during a contingency analysis run. When this box is not checked, no results will appear for the custom monitor following a contingency analysis run.

Choose Object

All Objects of Element Type

Select this option to monitor all objects of the selected **Element Type**.

Specific Object

Select this option to monitor only a specific object of the selected **Element Type**. When this option is selected, the dialog will be modified with a chooser box that will allow selection of a specific object.

Choose a Field

Select the field to be monitored for the selected **Element Type**. A field must be selected.

Choose a Pre Filter

Name of an [advanced filter](#) or [device filter](#). This is an optional field that can be left blank. If this is specified, an object must meet this filter in the contingency reference state in order for it to be monitored in the contingency analysis run. This filter can be used in conjunction with the option for monitoring **All Objects of Element Type** to select a specific set of objects rather than all objects of the selected **Element Type**.

Click the **Add/Modify** button to specify this filter.

Choose a Post Filter

Name of an [advanced filter](#) or [device filter](#). This is an optional field that can be left blank. If this is specified, an object must meet this filter in the post-contingency state in order for it to be monitored in the contingency analysis run.

Click the **Add/Modify** button to specify this filter.

Contingency Options: Contingency Definitions

These options are all available on the [Contingency Analysis dialog](#) under the [Contingency Options tab](#) under the Contingency Definitions grouping:

All Contingency Elements

Contains a combined list of all the contingency elements for all contingency records.

Contingency Blocks and Global Actions

A [contingency block](#) stores a list of contingency actions and has a name (or label) associated with it. A contingency block can then be called from a contingency record. This allows you to define a block of common actions you wish to have processed during several different contingencies, and then assign the block to each contingency instead of constantly redefining the same actions for each contingency. The **Contingency Block Elements** contains a combined list of all the contingency block elements for all contingency blocks.

A [global action](#) is an action that will automatically be processed as part of EACH contingency unless an Inclusion Filter is specified that allows defining with which contingencies the action will be included. Thus, if you have actions that are to be performed in every single contingency you define, you can insert it once in the global actions list, and Simulator will automatically use the defined action for every contingency it processes.

Model Conditions

Clicking this button allows you to define [model conditions](#). Model Conditions may be used as part of the Model Criteria defined with [Contingency Elements](#).

Model Filters

Clicking this button allows you to define [model filters](#). Model Filters may be used as part of the Model Criteria defined with [Contingency Elements](#). Generator Post-Contingency AGC

Model Expressions

Clicking this button brings up a list of the [model expressions](#). You may then insert a model expression by right-clicking on the list and choosing Insert. This will bring up the Define [Model Custom Expressions](#) dialog. Model Expressions may be used as part of Model Conditions.

Model Condition and Filter options

This section allows you to choose how assumptions about [Model Conditions](#) and [Model Filters](#) are made during the contingency analysis. Assumptions about the Model Conditions and Model Filters will impact how the [Model Criteria](#) are evaluated for contingency elements. You can specify to **Verify Model Conditions and Filters**, **Assume Model Conditions and Filters are TRUE**, or **Assume Model Conditions and Filters are FALSE**.

Contingency Options: Distributed Computing

The Distributed Contingency Analysis tool is available as an add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for more details.

These options are all available on the [Contingency Analysis Dialog](#) under the [Contingency Options Tab](#) under the Distributed Computing Grouping.

Distributed Computing is available for use with Contingency Analysis. In order to use distributed computing you must first configure a list of remote computers which can be utilized along with appropriate authentication information for those computers. The computer list and authentication information is common to all the distributed computing tools in Simulator and can be found in the [Simulator Options Dialog](#), or reached with the Distributed Computing Options button. They are described in [Distributed Computing Options](#).

The only two options specific to Contingency Analysis for Distributed Computing are as follows

Use Distributed Computing

Check this box to signify that when processing contingencies distributed computing should be used.

Number of Contingencies per Process

When distributed computing is used for contingency analysis, the entire list of contingencies to be solved will be split up into groups of contingencies. This setting specifies the number of contingencies to include in each group. During the distributed computing, Simulator will send a group of contingencies to each computer available as specified in the [Distributed Computing Options](#). As the remote computers complete a group, Simulator will send another grouping to that computer. This will continue until the contingency analysis results are completed.

Contingency Options: Miscellaneous

These options are all available on the [Contingency Analysis dialog](#) under the [Contingency Options tab](#) under the Miscellaneous grouping.

Always save results with the contingency list when you save it to a file

If this option is checked, the violation results for each contingency will also be stored in an auxiliary file when the list of contingencies is saved to an auxiliary file. This allows for recovering the results of a contingency analysis run for a case, without reloading and re-running all the contingencies saved in the file.

Save contingency analysis definitions/results in the case PWB file

If this option is checked, the contingency definitions and any processed results will be saved with the load flow case in the PWB file when you save the case.

Setting Reference Case when Contingency Analysis is Opened

These options determine how the reference case for the contingency analysis is treated each time you re-open the contingency analysis. The first time you open the contingency analysis tool for a particular power flow case during a Simulator session, Simulator stores the current state of the power flow case as the reference case for the contingency analysis results. (See [Contingency Case References - Defining the Reference State](#) for more information.) However, each subsequent time you open the contingency analysis dialog in the same Simulator session, Simulator needs to know how to set the reference state for the case. The choices are:

Always set reference case to the current case

This option will always assume that any changes you have made to the load flow case since the contingency analysis was last opened should be applied and will store the current state of the load flow as the new reference state for the contingency analysis.

Always use the existing contingency analysis reference case

This option will assume that the original reference case stored when the contingency analysis tool was originally launched should always be the reference state for the contingency analysis. This means that any changes that have been made to the load flow case since the initial launch of the contingency analysis will be lost, as the contingency analysis tool will reset the load flow state to the state stored with the first contingency analysis instance.

Prompt for which reference case to use (the current case or the pre-existing reference case) whenever the Contingency Analysis Form is opened

When this option is checked, you will always be prompted when you re-open the contingency analysis following the initial instance. You will then have the option to choose from one of the two previous settings, to either set the reference state to the current case or use the existing reference case currently stored with the contingency analysis tool.

Contingency Analysis: Results Tab

The Results tab of the [Contingency Analysis dialog](#) contains many options for viewing the results of a contingency analysis run.

- [View Results by Element](#) : navigate the list of limit violations by the element which is violated.
- View Results by Contingency : navigate the list of limit violations by the contingency which caused the violation. Essentially shows the same layout as seen on the [Contingencies Tab](#).
- Contingency Violations List : get a combined list of all contingency/limiting element pairs which results in a violation
- What Actually Occurred : Get a list of actions that occurred during the selected contingency. This is especially useful for identifying those actions conditional on Model Criteria were applied. It is also helpful when using the special [Open with Breakers](#) contingency actions because the list will indicate which breakers need to be opened to isolate the devices.
- Contingency Violation Matrices : Provides access to a dialog where a matrix of contingency versus limit elements can be created with entries showing the flow or voltage of the element to the respective contingency.
- [Text File Report Writing](#) : provides options for writing out a text file report
- [Summary](#) : a simple text summary showing the progress of the contingency analysis run.

Contingency Results: View Results By Element

These contingency results are all available on the [Contingency Analysis dialog](#) under the [Contingency Analysis: Results tab](#).

The View Results By Element category contains five sub-pages: Lines/Transformers, Buses, Interfaces, Nomogram Interfaces, and Custom Monitors. The information contained on each of the sub-pages provides an alternate method of viewing information similar to that contained on the [Contingencies tab](#). The individual pages show model objects (subject to area/zone/owner and advanced filters) only if they are associated with specific contingencies. For objects other than Custom Monitors, the user can select any model object on its respective page to see how many times a violation occurred on the device during a run of a set of contingencies. When a particular device is selected, the two pages at the bottom give the details of the analysis for the selected device.

Note: the information contained in the [Contingencies](#) and [Contingency Definition](#) sections of the View Results By Element page is object specific. The information present only pertains to contingencies that resulted in violations on the selected object.

Contingency Results: View Results By Element - Contingencies Section

The Contingencies section gives a list of all the contingencies that caused a violation on the selected device during the analysis. This display is very similar to the [Contingency Violations Display](#). While the Contingency Violations Display shows the *elements violated under the contingency*, this display shows the *contingencies that caused the violation*. If you then select one of the contingencies in this list, the [Contingency Definition section](#) displays the actions that took place during the selected contingency. Also, when you have selected a contingency from this list, you can click the **Show Other Violations** button. This will change your dialog to the contingencies tab and select the contingency you have selected, thus allow you to see other violations caused by this contingency.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** found under the Combined Tables button. Selecting this will display a dialog box giving the details of the actions applied during all contingencies and any actions that may have been skipped.

This page provides an easier way to check the contingency results when you are concerned with the results for a specific device in the system. The View Results By Element page provides a much easier tool for this kind of examination, as opposed to looking through each contingency on the Contingencies tab and trying to find each instance of a violation on the desired element.

Contingency Definition Display

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV*, *PTI File Format*, or *Label*.

Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** found on the Model Conditions page accessed from the [Contingency Definitions grouping](#) on the Options tab of the Contingency Analysis dialog. Model Criteria can be defined from the local menu (**CTGElement records > Define Model Criteria** option) on the Contingency Definition Display, from the **Add (Model Criteria)** button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** pages in the [Contingency Definitions grouping](#) on the Options tab of the Contingency Analysis dialog.

Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

Contingency Options Tab: Report Writing

These options are all available on the [Contingency Analysis Dialog](#) under the [Results Tab](#).

Simulator can produce a report that details the results of the contingency analysis. The Report Writing Tab allows you to control the content and appearance of the report. By default, the report will identify each contingency, whether or not it could be solved, and what violations resulted from it. By selecting options on this tab, you can include additional information in the report.

Optional Report Contents

Case Summary

The case summary prints the [Case Description](#) and then tallies the number of different power system components in the model.

Option Settings

If this item is checked, the report will list each of the options selected on the [Options Tab](#) of the Contingency Analysis Dialog.

Monitored Areas, Monitored Zones

If either of these items is checked, the report will identify the areas and/or zones in which Simulator has looked for limit violations, and over what voltage ranges.

Line Flow, Interface Flow, and Bus Voltage Extremes

If one of these items is checked, the report will list the worst-case line flows or voltages seen for each monitored element during the contingency analysis. None, one, two, or all three of these can be chosen at one time.

Base Case Outages

If this item is checked, the report will list the limit violations that existed in the Base Case.

All

Selecting this option will select all of the other options.

Identify buses by

This setting determines how the buses are listed in the data stored in the report. You can choose to have the buses displayed by number or name only, or by a combination of the number and name. You can also choose to identify with nominal voltage by checking the box labeled **Identify with Nominal Voltage**.

Show the actions involved in each contingency

If this box is checked, the definition of each contingency will be included in the report. The definition of each contingency simply identifies the actions that were implemented as part of the contingency.

Report only contingencies that cause violations

Checking this option will cause only the contingencies that cause violations to be shown in the report. Any contingency that did not cause a violation will not be included in the report. This option is useful if you wish to limit the size of the contingency report.

Report only limit type with violations for each contingency

Checking this option will result in reporting only the contingency violations of the type checked in the **Limit Type Violations to Include** box.

Report Inactive Violations and show all Rating Sets

When this box is checked, all violations that are normally being ignored during the contingency analysis (for example, base case violations) will be written to the report. Included with this option is the ability to show the different rating sets for each violated element.

Limit Type Violations to Include

Check the boxes of the types of violations you would like to be written to the report. This corresponds to the **Report only limit type with violations for each contingency** option above.

Maximum Violations of a single type to report

Enter the value of the maximum number of violations of a single type to be written to the report.

Create database-friendly tables

Checking this box will create three additional files for viewing contingency results: FILENAME_ctgelem, FILENAME_ctgviol, and FILENAME_ctgstat where FILENAME is the name of the file where the main report is saved. The user can choose which symbol to use to separate the columns by choosing a delimiting symbol. These files can be easily imported into a database or spreadsheet program such as Access or Excel as delimited text files, where they can be analyzed more rigorously. These files will be automatically created and saved in the same folder as the main report.

Produce Report

Clicking this button will open a Save Dialog for selecting the file to which the report should be written. The file is saved as a text file (*.txt). Enter the file path and name and click **Save** to write the report to the selected file. After saving the file, a dialog will provide the option of viewing the report file immediately.

Contingency Results: Summary

These options are all available on the [Contingency Analysis Dialog](#) under the [Results Tab](#).

The Summary Tab of the [Contingency Analysis Dialog](#) provides additional information on the status of the contingency analysis run and allows you to start, pause, resume, and abort the contingency run. The top half of the Summary Tab charts the progress of the contingency analysis run and issues warning messages when a particular contingency fails to solve. The next section features counters that indicate the total number of contingencies that comprise the list, the number of these contingencies that have been processed thus far, the number of contingencies that failed to solve, and the total number of violations that have been flagged.

Comparing Contingency Analysis Results

After performing an automated contingency analysis, the results of the analysis can be saved to an auxiliary file. Once the results have been saved to a file, they can then be compared to different results in another file or to results existing in memory in the Contingency Analysis tool. To compare two sets of contingencies, do the following:

- Process each set of contingencies and save the results for at least one set in an auxiliary file.
 - To save a contingency list, right-click on the list of contingencies and choose **Save As > Auxiliary File**.
 - In the Save Contingency File Dialog, choose a name for the file, and then click **Save**.
 - You will then be prompted to choose options for saving the contingency list to an auxiliary file. By default, the contingency definitions themselves are saved, along with the contingency options. The optional information you may choose to save are the [Limit Monitoring Settings](#), General [Power Flow Solution Options](#), [List Display Settings](#), and [Contingency Results](#). When saving the contingency results with the file, you may also choose to include inactive violations.

Inactive violations are considered violations on elements for a DIFFERENT limit than what is currently being monitored. For example, a branch may have an A limit rating of 50 MVA, and a B limit rating of 100 MVA. Consider if the B limits are being used to report violations during a contingency. A value of 75 MVA flow on the branch would not be reported in the contingency analysis as a violation, considering the B limit of 100 MVA is being used. However, Simulator internally will flag the element as a potential violation if the limit set used is switched to the A rating set. Simulator considers these types of situations as inactive violations. These are kept track of to allow the user to easily switch the rating set used for reporting violations from one set to another and see the results immediately, without having to re-run the entire contingency set to determine the violations for the new rating set. Choosing to include the inactive violations when saving an auxiliary file maintains this flexibility when the contingency definitions and results are read into a case from an auxiliary file. You can also choose for the identifiers used in the file to be either the bus numbers or the bus name and nominal kV voltage.

Note: For comparing two lists of contingency results, you MUST save the contingency results with each of the two auxiliary files being compared.
 - Click **OK** to save the contingencies and the results to the auxiliary file specified.
- Once you have two different lists of contingency results (at least one of the lists must be saved in an auxiliary file), right-click on the contingency list and choose **Compare Two Lists of Contingency Results** or choose **Other > Compare Two Lists of Contingency Results**. This will bring up a dialog on which you have to specify the Contingency Lists you are interested in comparing. You must specify the **Controlling Contingency List** and the **Comparison Contingency List**. The definitions of these two lists are found below.
- Click on the **Browse** buttons to specify the two Contingency Lists you would like to compare. You can also choose to use the presently open Contingency List as either the Controlling Contingency List or the Comparison Contingency List.

Controlling Contingency List:

The list that controls what is displayed on the dialog. Only contingencies that are defined in this list will be displayed on the form. Only violations that occur for contingencies in this list will appear in the Violations List for each contingency.

Comparison Contingency List:

This is the list to which the Controlling Contingencies will be compared. Comparisons will occur for those contingencies in the lists that have the same **CONTINGENCY NAME**. Note that contingencies in the Comparison list whose **CONTINGENCY NAME** does not match one of those in the Controlling list will not be displayed. Also, violations which occur in a specific Comparison contingency that do not occur in the respective Controlling contingency will not be displayed.

Example:

A user has a power system case and a list of contingencies. The user runs contingency analysis on this system for this list of contingencies. The results are saved in a file called comparison.aux. The user now changes the system state, possibly adding in a 500 MW transaction between two areas. The contingency analysis is run on this new state of the system for the list of contingencies. The results are saved in a file called controlling.aux. You should define the contingency results you are more interested in viewing as the Controlling List because this list determines what is shown on the dialog. In this case, we are more interested in seeing the violations caused when the transaction is in place, so that list is defined as the Controlling List.

The comparison of the two sets of contingencies is now done by right-clicking and choosing **Compare Two Contingency List Results**. The file controlling.aux is set as the **Controlling List** and comparison.aux as the **Comparison List**.

- After clicking **OK** on the dialog, the contingency lists will be read from the specified files or from the presently open list. After Simulator has completed reading these files, a prompt will appear which asks, "**Would you like to set the dialog with default columns for comparing contingency lists?**". It is recommended that you choose **YES** so that the case information displays on the Contingency Dialog will automatically be set to show fields that will help you compare the two lists of contingencies. For information on the default fields used when comparing contingencies see [Comparing Contingencies List Displays](#).

Comparing Contingencies List Displays

When comparing two sets of contingency results, there are several additional default fields added to several list displays which help you compare the results. These fields are described below for the various list displays.

Contingencies Records on [Contingencies Tab](#)

Violations

The total number of violations for this contingency under the Controlling List.

Comp Violations

The total number of violations for this contingency under the Comparison List.

New Violations

The number of violations which occur in the Controlling List which do not occur in the Comparison List.

Max Branch % (Max Interface %)

The highest branch (interface) violation caused by this contingency in the Controlling List.

Comp Max Branch % (Comp Max Interface %)

The highest branch (interface) violation caused by this contingency in the Comparison List.

Worst Branch Violation (Worst Interface Violation)

This is the maximum of the following two values: [**Worst Increase Violation**] and [**Worst New Violation - 100%**]

Min Volt (Max Volt)

The worst violation in the controlling list.

Comp Min Volt (Comp Max Volt)

The worst violation in the comparison list.

Worst LowV Violation (Worst HighV Violation)

The worst new violation.

[Contingency Violations Display](#) on [Contingencies Tab](#) and Contingency Violations on the [View Results By Element Page](#)

When Viewing Lines/Transformers, Buses, Interfaces, or Nomogram Interfaces

Value

The value of the violation under the controlling list.

Comp Value

The value of the violation under the comparison list.

Diff Value

The difference between **Value** and **Comp Value**

Limit

The limit of the element in the controlling list.

Comp Limit

The limit of the element in the comparison list.

Diff Limit

The difference between **Limit** and **Comp Limit**

Percent

The percent violation in the controlling list.

Comp Percent

The percent violation in the comparison list.

Diff Percent

The difference between **Percent** and **Comp Percent**

Contingency Violations on the [View Results By Element Page](#) When Viewing Custom Monitors

Value

The value of the monitor under the controlling list.

Reference Value

The value of the monitor in the reference state under the controlling list.

Change Value

The difference between **Value** and **Reference Value**

Comp Value

The value of the monitor under the comparison list.

Diff Value

The difference between **Value** and **Comp Value**

Line/Transformer (Interface) Records on the [View Results By Element Page](#)

Violations

This shows the number of branch (interface) violations which occurred in the Controlling List

New Violations

This shows the number of branch (interface) violations which occurred in the Controlling List, but did not occur in the Comparison List.

Max % Loading Cont.

The worst branch (interface) violation in the controlling list.

Max % Ld Cont Comp

The worst branch (interface) violation which occurred in the comparison list.

Worst Increased Violation

The worst increase in a branch (interface) violation from the comparison list to the controlling list.

Bus Records on the [View Results By Element Page](#)

Violations

This shows the number of violations which occurred in the Controlling List

New Violations

This shows the number of violations which occurred in the Controlling List, but did not occur in the Comparison List.

Max Voltage Cont.

The worst high voltage violation in the controlling list.

Max Voltage Cont Comp

The worst high voltage violation which occurred in the comparison list.

Worst Max Volt CTG Change

The worst increase in a high voltage violation.

Min Voltage Cont.

The worst low voltage violation in the controlling list.

Min Voltage Cont Comp

The worst low voltage violation which occurred in the comparison list.

Worst Min Volt CTG Change

The worst decrease in a low voltage violation.

Nomogram Interfaces Records on the [View Results By Element Page](#)

Violations

This shows the number of violations which occurred in the controlling list.

Max % Loading Cont.

The worst nomogram interface violation in the controlling list.

Custom Monitor Records on the [View Results By Element Page](#)

No changes.

Contingency Element Dialog

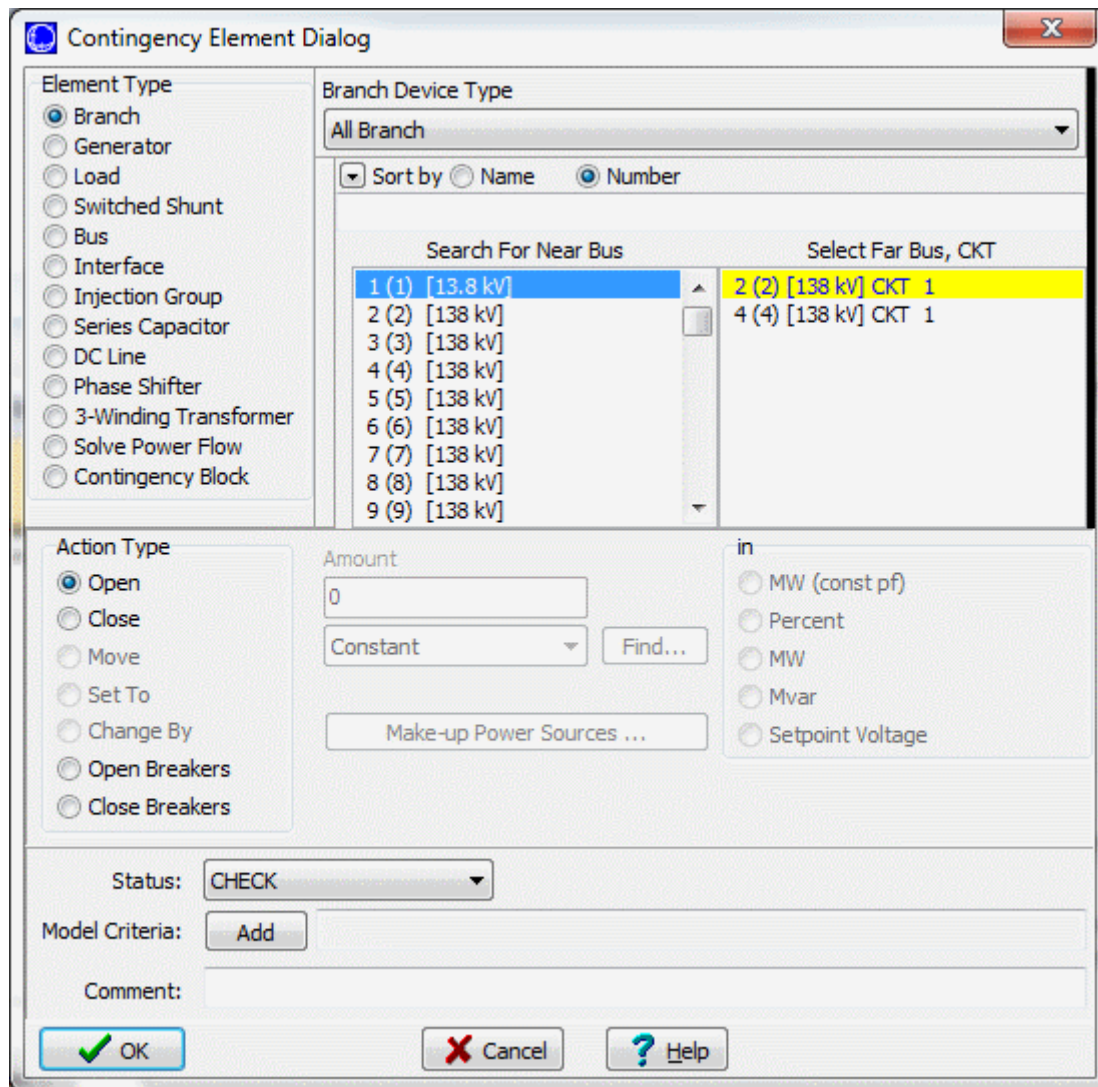
The Contingency Element Dialog provides information on the individual elements that comprise a contingency definition. You may use this dialog to modify an existing contingency's definition or to add elements to new or existing contingencies.

There are several ways to open the Contingency Element Dialog:

- By pressing the **Insert New Element** button on the [Contingency Definition Dialog](#).
- By right-clicking on a [Contingency Definition Display](#) and choosing **Insert** or **Show Dialog**.
- By right-clicking on the [Contingency Tab](#) of the [Contingency Analysis Dialog](#) and choosing **Insert Special > Quick Insert of Single Element Contingency**.

After making the desired changes, click **OK** to save changes and close the dialog or click **Cancel** to close the dialog without saving your changes.

Click **Delete** to remove the element from the contingency.



Contingency Element Dialog

The Contingency Element Dialog has the following controls:

Element Type

Indicates whether the element involved in the contingency action is a branch, generator, load, switched shunt, bus, interface, injection group, series capacitor, DC line, phase shifter, [contingency block](#) or solve power flow. The Element Type will dictate what **Action Types** are available, and the selection of the Element Type, in conjunction with the **Action Type**, **Amount**, and **in** options settings, determines what actually happens during the contingency.

What actually happens during a contingency action based on the **Element Type**, **Action Type**, and **in** options is described in detail below:

Branch

If **Branch Device Types** other than *Line* exist in the case, a special filtering drop down will appear that allows selecting branches by device type. Selecting a particular type will limit the entries in the chooser display to branches of that type.

Open

The Open action will set the Status of the selected branch to *Open* if the branch Status is *Closed*. If the branch Status is already *Open*, this action does nothing.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of the selected branch to *Closed* if the branch Status is *Open*. If the branch Status is already *Closed*, this action does nothing.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Generator

Open

The Open action will set the Status of the selected generator to *Open* if the generator Status is *Closed*. If the generator Status is already *Open*, this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of the selected generator to *Closed* if the generator Status is *Open*. If the generator Status is already *Closed*, this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Move

The Move action allows the transferring of generation from an existing generator bus to another bus. A generator move can be done on a bus basis (impacting the total generation of all generators at the selected bus) or can be targeted at a specific generator. Generators that are adjusted by the Move action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation move will occur. Generation may be moved in **Percent** or **MW**. A **Percent** move will move the specified percent of the current generator MW and Mvar to the defined **Bus to Move to**. A **MW** move will move the specified amount of generator MW to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the output of the selected generator bus, and a negative amount will increase the output of the selected generator bus. The Bus to **Move to** will respond accordingly.

If a generator already exists at the bus to which the generation is being moved, the generation at that bus is adjusted according to the move amount. If a generator does not exist but a load exists, then the load is adjusted by the move amount. If no generator or load exists, then a load is added at the bus and the injection is set according to the move amount. In terms of power injection, positive generation is the same as negative load.

Set To

The Set To action will set the selected bus' generation or specified generator's parameters to a specified **Amount**. Generator Set To actions can be done on a bus basis (impacting the total generation of all generators at the selected bus) or can be targeted at a specified generator. Generators that are adjusted by a Set To action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be set in **Percent**, **MW**, or **Setpoint Voltage**. When the generation is set in **Percent**, total generator MW output will be set to a value based on the specified percent of the current total generator MW output. When the generation is set in **MW**, total generator MW output will be set to the specified MW amount. When the generation is set in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be set to the specified amount in per unit.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Change By

The Change By action will change the selected bus' generation or specified generator's parameters by a specified **Amount**. Generator Change By actions can be done on a bus basis (impacting the total generation of all generators at the selected bus) or can be targeted at a specified generator. Generators that are adjusted by a Change By action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be changed in **Percent**, **MW**, or **Setpoint Voltage**. When the change is in **Percent**, total generator MW output will be changed based on the specified percent of the total current generator MW output. When the change is in **MW**, total generator MW output will be changed by the specified MW amount. When the change is in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be changed by the specified amount in per unit.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

Load

Open

The Open action will set the Status of the selected load to *Open* if the load Status is *Closed*. If the load Status is already *Open*, this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of the selected load to *Closed* if the load Status is *Open*. If the load Status is already *Closed*, this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Move

The Move action allows the transferring of load from an existing load bus to another bus. A load move can be done on a bus basis (impacting the total load of all loads at the selected bus) or can be targeted at a specific load. Load may be moved in **MW (const pf)**,

Percent, MW, or Mvar. The load amount that is moved is the total actual load and may be different than the nominal load if the area of the load has a Load MW Multiplier other than 1.0 or the load contains constant impedance or constant current components. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. A **MW (const pf)** move will move the specified amount of load MW to the **Bus to Move to** while moving a Mvar amount that will maintain a constant power factor at the selected load bus. A **Percent** move will move the specified percent of the present load MW and Mvar to the defined Bus to Move to. A **MW** move will move the specified amount of the load MW to the Bus to Move to. A **Mvar** move will move the specified amount of the load Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the load at the selected bus, and a negative amount will increase the load at the selected bus. The Bus to **Move to** will respond accordingly.

If a load already exists at the bus to which the load is being moved, the load at that bus is adjusted according to the move amount. If no load exists, then a load is added at the bus and the load is set according to the move amount.

Set To

The Set To action will set the selected bus' load parameters to a specified **Amount**. Load Set To actions can be done on a bus basis (impacting the total load of all loads at the selected bus) or can be targeted at a specific load. Load parameters can be set in **MW (const pf), Percent, MW, or Mvar**. The load value that is set for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus and the bus at which the load is located is *Closed*, load status at the selected load bus will change so that one load is *Closed* and the total constant power load at the bus will be set to the specified amount. If the bus at which the load is located is *Open* or there is no existing load, either *Open* or *Closed*, at that bus, then no load change will occur. When the load is set in **MW (const pf)**, the actual constant power MW will be set to the specified amount and the actual constant power Mvar will be set so that a constant power factor is maintained at the load bus. When the load is set in **Percent**, the actual constant power MW and Mvar will be set to the specified percent of the present load MW and Mvar. When the load is set in **MW**, the actual constant power MW at the selected load will be set to the specified amount. When the load is set in **Mvar**, the actual constant power Mvar at the selected load will be set to the specified amount.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Change By

The Change By action will change the selected bus' load parameters by a specified **Amount**. Load Change By actions can be done on a bus basis (impacting the total load of all loads at the selected bus) or can be targeted at a specific load. Load parameters can be changed in **MW (const pf), Percent, MW, or Mvar**. The load value that is changed for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. When the load is changed in **MW (const pf)**, the actual constant power MW will be changed by the specified amount and the actual constant power Mvar will be changed so that a constant power factor is maintained at the load bus. When the load is changed in **Percent**, the actual constant power MW and Mvar will be changed by the specified percent of the present load MW and Mvar. When the load is changed in **MW**, the actual constant power MW at the selected load bus will

be changed by the specified amount. When the load is changed in **Mvar**, the actual constant power Mvar at the selected load bus will be changed by the specified amount.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

Switched Shunt

Open

The Open action will set the Status of the selected switched shunt to *Open* if the switched shunt Status is *Closed*. If the switched shunt Status is already *Open*, this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of the selected switched shunt to *Closed* if the switched shunt Status is *Open*. If the switched shunt Status is already *Closed*, this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Move

The Move action allows the transferring of switched shunt MW and/or Mvar from an existing switched shunt bus to another bus. A switched shunt move can be done on a bus basis (impacting the total shunt output of all switched shunts at the selected bus) or can be targeted at a specific switched shunt. During the move action, the nominal values of the switched shunts are adjusted, and switched shunts that are adjusted have their Control Mode set to *Fixed*. If there are no switched shunts at the selected switched shunt bus with a Status of *Closed*, no switched shunt move will occur. Switched shunt Moves may be done in **Percent**, **MW**, or **Mvar**. A switched shunt Move in **Percent** will move the specified percent of present nominal MW and Mvar at the switched shunt bus to the defined **Bus to Move to**. A **MW** move will move the specified amount of switched shunt MW to the Bus to Move to. A **Mvar** move will move the specified amount of switched shunt Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the nominal MW and/or Mvar at the selected switched shunt bus, and a negative amount will increase the nominal MW and/or Mvar at the selected switched shunt bus. The Bus to Move to will respond accordingly.

If a switched shunt exists at the bus to which the switched shunt MW and/or Mvar is being moved, the switched shunt at that bus is adjusted according to the move amount. If there are existing switched shunts at the Bus to Move to but all of them have a Status of *Open*, the first switched shunt Status will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal MW and/or Mvar will be adjusted according to the move amount. If no switched shunt exists at the Bus to Move to, then the nominal constant impedance load is adjusted according to the move amount. If no switched shunt exists at that bus, then a switched shunt is added and the nominal injection is adjusted according to the move amount.

Set To

The Set To action will set the selected bus' switched shunt parameters to a specified **Amount**. Switched shunt Set To actions can be done on a bus basis (impacting the total shunt output of all switched shunts at the selected bus) or can be targeted at a specific switched shunt. Switched shunts that are adjusted during Set To actions have their Control Mode set to *Fixed*. Switched shunt parameters can be set in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is set in **Percent**, the nominal MW and Mvar are set to the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is set in **MW**, the nominal MW is set to the specified amount. When the switched shunt bus is set in **Mvar**, the nominal Mvar is set to the specified amount. When the switched shunt bus is set in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is set to the specified amount. Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amounts, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

Change By

The Change By action will change the selected bus' switched shunt parameters by a specified **Amount**. Switched shunt Change By actions can be done on a bus basis (impacting the total shunt output of all switched shunts at the selected bus) or can be targeted at a specific switched shunt. Switched shunts that are adjusted during the Change By action have their Control Mode set to *Fixed*. Switched shunt parameters can be changed in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is changed in **Percent**, the nominal MW and Mvar are changed by the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is changed in **MW**, the nominal MW is changed by the specified amount. When the switched shunt bus is changed in **Mvar**, the nominal Mvar is changed by the specified amount. When the switched shunt bus is changed in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is change by the specified amount. Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage change is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amount, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

Bus

Open

The Open action will set the Status of all ac branches connected to the selected bus to *Open*. If all of the ac branches are already open, then this action does nothing.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Interface

Open

The Open action will set the Status of all ac branches in the selected interface to *Open*. If all of the ac branches are already open, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Open*.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of all ac branches in the selected interface to *Closed*. If all of the ac branches are already closed, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Closed*.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Injection Group

Open

The Open action will set the Status of all generators and loads in the selected injection group to *Open* if the Status is *Closed*. If the Status is already *Open*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Open*.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of all generators and loads in the selected injection group to *Closed* if the Status is *Open*. If the Status is already *Closed*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Closed*.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Set To

The Set To action will set the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent** or **MW**. When setting the injection in **Percent**, the real power injection is set to the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When setting the injection in **MW**, the real power injection is set to the specified amount.

A checkbox is also available which says **Use Merit Order For Generators**. Normally when not checking this box all generators and loads in the injection group are adjusted according to

their relative participation factors and thus all with non-zero participation factors will be adjusted to meet the desired injection. When checking the option **Use Merit Order for Generators**, only generators will be adjusted to meet the desired injection. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum MW limit before moving on to the next generator. This process continues until the desired injection is met. Note that generators will not be changed opened in the process so all generators will continue to provide Mvar support.

If **Use Merit Order For Generators** is checked, another checkbox labeled **Open Generators in Merit Order** will be available. If this option is chosen and the MW output being requested is *lower* than the existing MW output of the generators in the injection group, then the merit order dispatch will be modified. (Note: PowerWorld's expectation is that the Change By option would most frequently be used with this option.) In this case, the generator in the injection group with the highest participation factor will have its status changed to Open, followed by the second generator and so on. This will continue until the amount of MW opened is as close to the desired amount as possible but has *not exceeded* the desired amount of change. If opening a generator will cause the amount of MW opened to exceed the desired amount, that generator will be skipped and the next one in merit order will be examined. If the MW output requested is higher than the present MW output, then generators will be increased toward their maximum MW output in the same manner as though this option was not chosen.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Change By

The Change By action will change the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent** or **MW**. When setting the injection in **Percent**, the real power injection is set to the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When setting the injection in **MW**, the real power injection is set to the specified amount.

A checkbox is also available which says **Use Merit Order For Generators**. Normally when not checking this box all generators and loads in the injection group are adjusted according to their relative participation factors and thus all with non-zero participation factors will be adjusted to meet the desired injection. When checking the option **Use Merit Order for Generators**, only generators will be adjusted to meet the desired injection. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum MW limit before moving on to the next generator. This process continues until the desired injection is met. Note that generators will not be changed opened in the process so all generators will continue to provide Mvar support.

If **Use Merit Order For Generators** is checked, another checkbox labeled **Open Generators in Merit Order** will be available. If this option is chosen and if the MW output change being requested is *negative*, then the merit order dispatch will be modified. In this case, the generator in the injection group with the highest participation factor will have its status changed to Open, followed by the second generator and so on. This will continue until the amount of MW opened is as close to the desired amount as possible but has *not exceeded* the desired amount of change. If opening a generator will cause the amount of MW opened to exceed the desired amount, that generator will be skipped and the next one in

merit order will be examined. If the change requested is positive, then generators will be increased toward their maximum MW output in the same manner as though this option was not chosen.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

Series Capacitor

Bypass

The Bypass action will change the Bypass Status of the selected series capacitor to *Bypassed* if the Bypass Status is *Not Bypassed*. If the Bypass Status is already *Bypassed*, this action does nothing.

Inservice

The Inservice action will change the Bypass Status of the selected series capacitor to *Not Bypassed* if the Bypass Status is *Bypassed*. If the Bypass Status is already *Not Bypassed*, this action does nothing.

Set To

The Set To action will set the series reactance of the series capacitor to a specified **Amount**. The reactance may be set in **X (percent)** or **X (per unit)**. If the reactance of the series capacitor is set in **X (percent)**, the per unit reactance is set to the specified percent of the present per unit reactance of the series capacitor. If the reactance is set in **X (per unit)**, the per unit reactance of the series capacitor is set to the specified amount.

DC Line

Open

The Open action will set the Control Mode of the selected dc line to *Blocked*. If the dc line is already blocked, this action does nothing.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Control Mode of the selected dc line to either *Power* or *Current*. When selecting the Close action, the **Amount** must be specified for the new Setpoint of the dc line in either **MW** or **Amps**. If specifying the Setpoint in **MW**, the Control Mode of the dc line will be set to *Power*. If specifying the Setpoint in **Amps**, the Control Mode of the dc line will be set to *Current*.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Set To

The Set To action will set the selected dc line's Setpoint to a specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be set in **Percent**, **MW**, or **Amps**. When the dc line's Setpoint is set in **Percent**, the Setpoint is set to the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is set in **MW**, the Setpoint is set to the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is set in **Amps**, the Setpoint is set to the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, then the Control Mode remains unchanged.

Change By

The Change By action will change the selected dc line's Setpoint by the specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be changed in **Percent**, **MW**, or **Amps**. When the dc line's Setpoint is changed in **Percent**, the Setpoint is changed by the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is changed in **MW**, the Setpoint is changed by the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is changed in **Amps**, the Setpoint is changed by the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, the Control Mode remains unchanged.

When the Control Mode changes from either *Power* to *Current* or from *Current* to *Power*, the appropriate conversions are done so that the resulting Setpoint value reflects the original Setpoint and the change in Setpoint in the same units.

Phase Shifter

Set To

The Set To action will set the middle of the regulation range of the selected phase shifter to the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and Regulation Maximum MW Flow. The middle of the regulation can be set in **Percent** or **MW**. If the middle of the regulation range is set in **Percent**, the middle of the range is set to the specified percent of the present middle of the regulation range. If the middle of the regulation range is set in **MW**, then the middle of the regulation range is set to the specified amount. Regardless of the method used to set the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.

Change By

The Change By action will change the middle of the regulation range of the selected phase shifter by the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and the Regulation Maximum MW Flow. The middle of the regulation range can be changed in **Percent** or **MW**. When the middle of the regulation range is changed in **Percent**, the change is based on the specified percent of the present middle of the regulation range. When the middle of the regulation range is changed in **MW**, the middle of the present regulation range is changed by the specified amount. Regardless of the method used to change the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.

3-Winding Transformer

Open

The Open action will set the Status of the selected three-winding transformer to *Open* if the three-winding transformer Status is *Closed*. If the three-winding transformer Status is already *Open*, this action does nothing.

Open Breakers

See the special topic on [Open Breakers](#) for more detail.

Close

The Close action will set the Status of the selected three-winding transformer to *Closed* if the three-winding transformer Status is *Open*. If the three-winding transformer Status is already *Closed*, this action does nothing.

Close Breakers

See the **Close Breakers for Contingencies** topic below.

Solve Power Flow

The solve power flow type is unique in that you can include an action that forces Simulator to solve the power flow as part of the contingency. There are rare special cases of sophisticated contingency definitions this can be used for, as requested by one or more PowerWorld customers. The load flow is already generally solved for each contingency as part of the processing.

If a contingency has one or more Solve Power Flow actions, then there will be a few changes in how the contingency definition display behaves.

- Sorting of the list of actions is no longer allowed. This is because the order of the actions is now important to how the contingency is processed.
- When you right-click on the list of contingency actions there will be two new options for Move Up and Move Down. These can be used to reorder the actions.
- On the Contingency Definition Dialog, there will be up/down arrows on the right of the dialog that may be used to reorder the actions.

Contingency Block

The individual actions of the Contingency Block are applied according to how they are defined. There are no Action Types available for a Contingency Block.

Choose the Element

Use this portion of the dialog to choose the element involved in this action. This behaves the same as the [Advanced Find Dialogs](#) used throughout the software.

Action Type

Defines the change specified by the contingency action. The Action Types available depend on the **Element Type** selected. Possible Action Types include: Open, Close, Move, Set To, Change By, Bypass, and Inservice. The behavior of each of these actions for the different elements is described with the **Element Types**.

Amount

Enterable fields used to specify the quantity of change desired for the contingency element. The Amount fields are enabled when the **Action Type** is set to *Move*, *Set To*, or *Change By*. The availability of these Action Type options changes depending on the **Element Type** selected. The top field is used to define a *Constant*, *Field*, or [Model Expression](#) depending on the entry selected in the bottom drop-down box. Use the **Find** button to display a dialog listing *Fields* or *Model Expressions*. This dialog behaves the same as the [Advanced Find Dialogs](#) used throughout Simulator.

The Amount of change that occurs is based on the value of the top field entry. The top entry field is a constant, field name, or model expression name. When this entry is a field name, then the value of the change is based on the value of the selected element field when the contingency occurs. When this entry is a model expression, then the value of the change is based on the value of the selected model expression when the contingency occurs.

The Amount of change that occurs for the selected element is based on the value entered in the top field and the parameter set with the **in** options. The availability of the **in** options will change based on the Element Type selected and the Action Type selected. The availability of these options is described in detail with the **Element Types**.

Make-up Power Sources

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting real power imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

Model Criteria

Click the **Add** button to specify a criterion under which the contingency action will occur. Either a [Model Filter](#) or [Model Condition](#) can be selected for the Model Criteria. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. This condition should be specified as a Model Condition and then this condition selected as the Model Criteria. Normally, no Model Criteria will be specified for a given action and this field will be blank.

Comment

An optional user-specified comment string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#) or with the case PWB file.

Close Breakers for Contingencies

When using the **Close Breakers** contingency action, breakers are identified individually for each element that is using this action and the element is not already energized. A list of breakers to be closed is created for each affected element regardless of whether closing the breakers will actually energize the element. The lists of breakers for each individual element are merged into a set of unique breakers. New temporary contingency actions are created for these breakers so that when the contingency action is actually implemented, the selected breakers are closed instead of the element itself changing status. Once the contingency has been implemented and solved, the temporary contingency actions are removed so that the user never sees the actions created for the breakers. During the actual contingency solution, the *What Actually Occurred?* list will be updated and can be examined in the results to determine what breakers were needed to energize the device, even though the actions created for the breakers are removed after the contingency has been examined.

The **Close Breakers** contingency action is only available for contingency records and is not available for [contingency block definitions](#) or for [global contingency actions](#).

More information about how breakers are selected can be found in the [Close Breakers Overview](#) topic.

Contingency Element: Open Breakers

The contingency action type *Open Breakers* is available for each contingency element type for which the *Open* action type is available. This includes the following element types: Branch, Bus, Load, Generator, Switched Shunt, DC Line, Injection Group, and Interface. This action type is only available for contingency records and is not available for [contingency block definitions](#) or for [global contingency actions](#).

In order to handle the *Open Breakers* contingency elements, before one contingency's solution is performed, the list of contingency elements for that contingency is processed by looking for elements which use *Open Breakers*. For elements with this action type, all the terminal buses of the respective device are processed using the algorithm below to determine which breakers to open to isolate the device (and possible which generators to outage). The basic idea of the algorithm is to search for a list of breakers that will isolate all the terminal buses of the device from in-service generation. The algorithm will also return a list of online generators which must be opened to isolate the bus as well. Remember that a breaker is specified by setting the **Branch Device Type** to *Breaker*.

Using the list of breakers and generators found from the algorithm, contingency analysis augments the contingency element list with a dynamically created list of contingency elements of action type *Open* for each breaker and generator device. The dynamically created elements are given the same [Model Criteria and Status](#) as the contingency element which required their creation. Note that if multiple *Open Breakers* contingency elements require the same breaker to open, then one new contingency element will be created. When *Open Breakers* contingency elements have different Model Criteria, then a new contingency element will be created for each unique Model Criteria. During the actual contingency solution, the *What Actually Occurred?* list will be updated and can be examined in the results to determine what breakers were needed to isolate the device. Finally, after the contingency solution is completed, all the dynamically created contingency elements are deleted, and as a result the user can only see which breakers or generators were opened by looking at the *What Actually Occurred* list.

For brevity in the context of tools that either automatically switch or identify breakers to close a device, the term *breaker* is used to indicate any type of device that can be switched automatically. This includes **Branch Device Types** *Breaker* and *Load Break Disconnect*. If the model has no branches defined with either of these Branch Device Types, this option will not work because the algorithm will never find any breakers. See the [Full Topology Model](#) topic for more information on defining breakers.

Algorithm description for Open with Breakers

This algorithm is applied to each terminal bus of the device specified. For generators, loads, shunt, and buses this is done on a single bus only. For an Injection Group however, each generator, load, and shunt in the injection group will be processed in turn. Starting at each terminal bus of the device and marching outwards, the algorithm traverses closed branches (except for breakers), dc lines, and multi-terminal DC lines and will flag any closed breakers at buses which are visited. The buses that are visited are also flagged. (Note: the algorithm will NOT traverse across the breakers with for an exception noted below). When buses are visited that have online generators, a list of generators is maintained that must be opened as well. In order to prevent the degenerate case where breakers do not fully isolate any portion of the network, only 10 buses with online generation are permitted for visitation. If more than 10 buses are encountered with online generation, then the algorithm will immediately abort and a log message is written saying that the device "cannot be isolated from online generation".

If this part of the algorithm completes it will have obtained a list of flagged breakers and a list of visited buses (as well as a list of generators to open). The list of flagged breakers is then examined to ensure that each breaker actually isolates the original device. Each flagged breaker will be examined and if a flagged breaker has a terminal bus that is NOT in the list of visited buses, then

this breaker disconnects the device from another part of the system and will be maintained. Breakers for which both terminal buses have been visited will be discarded because these breakers do not actually isolate anything. If at least one flagged breaker is maintained then the process is deemed successful and the contingency will outage the list of maintained flagged breakers. If all the flagged breakers are discarded, then the breakers do not isolate anything, and therefore a log message is written saying that "no part of the system can be isolated," and nothing is done.

A few special notes:

1. If there is another contingency action inside this same contingency that specifies an action type of *Open* or *OpenCBs* for a generator device, encountering that generator won't count toward the list of generators to open or the maximum of 10 buses to encounter with online generation. The same is done for any generator connected to a bus or inside an injection group, if the bus or injection group has a contingency action of type *Open* or *OpenCBs*.
2. If there is a contingency action inside this same contingency that specifies a breaker with action type *Open Breakers*, the algorithm will traverse across this breaker (this models a breaker failure where a breaker is supposed to open but fails to operate.)
3. If there is a contingency action inside this same contingency which specifies an action type of *Open* for a branch or DC line device, the algorithm will NOT traverse across that branch or DC line. The same is done for any branch which is part of a three-winding transformer, interface, or connected to a bus if the three-winding transformer, interface, or bus has a contingency action of type *Open*.
4. While a list of generators to open is maintained, if no online load is encountered while traversing buses in the algorithm, then the generator OPEN contingency actions will not be created. This is because the breakers will create an isolated island with no load and thus opening the generation is not necessary.

Make-Up Power Sources

Generator, load, injection group, and switched shunt contingencies may cause imbalances between generated power and demand. The default way to handle these imbalances is to assign them to the system slack. However, this is probably unrealistic and, depending on the size of the unit or load involved, convergence problems may result.

To implement more local compensation for generation, load, injection group, and switched shunt changes, use the Make-Up Power Sources Dialog. This dialog can be accessed by from the [Contingency Element Dialog](#) by clicking the button labeled **Make-Up Power Sources**. Most of the dialog is occupied by a grid that lists bus numbers and relative contributions. This grid is a [Case Information Display](#), so its behavior should be familiar. For example, right-click on the grid to display its local menu.

We shall call those buses that must compensate for the changes caused by a generator, load, injection group, or switched shunt contingency "compensators". To insert a new compensator, select **Insert** from the grid's local menu. This opens another dialog, where you should specify the bus number of the compensator and its contribution. Specify the contribution of the compensator either as a percentage or as a fixed number of MW. If a particular contingency has multiple compensators, then the choice of basis for the contribution (either MW or percent) should be consistent for each. After identifying the compensator, click **OK**. The grid should update with your newly added compensator.

To delete an existing compensator, select it from the grid and select **Delete** from the local menu.

To determine how much each compensator contributes to the imbalance, each contribution value is normalized to the total sum of all contribution values. This allows the values to be entered in either percent or MW. Keep in mind that when entering the values in percent that the sum of all contribution values should equal 100 or the actual contribution amounts will not occur in the expected percentages.

Compensators account for the changes caused by a generator, load, injection group, or switched shunt contingency by changing either generation or load to satisfy its defined contribution. For example, suppose compensator contributions are specified as percentages, and Simulator needs to compensate for 100 MW lost in a particular generation contingency. Suppose the contingency has 4 compensators defined as follows:

Bus Number Contribution

1	20
2	30
3	10
4	40

Suppose buses 1 and 2 are load buses, and buses 3 and 4 are generators. Then bus 1's load will decrease by 20 MW, bus 2's load will decrease by 30 MW, bus 3's generation will increase by 10 MW, and bus 4's generation will increase by 40 MW. These changes will be instituted regardless of the compensators' operating limits or AGC status. In fact, any compensating units will be set off AGC to ensure that the prevailing AGC control does not distort the dictates of the contingency. Furthermore, maximum MW limits on generators will not be checked.

If a compensator has both generators and loads attached to it, the generator will take precedence. The generator will function as the compensating device, and the load will be left unchanged.

Contingency Make-Up Sources Dialog

Simulator allows you to specify buses that will serve as compensators to account for changes in the balance between generation and demand caused by generator and load contingencies. This dialog allows you to define or adjust the amount of power supplied by a compensator. For more information, please see [Make-Up Power Sources](#).

Tutorial: Contingency Analysis

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

1 2 3 4 5 6 7 8 9 10 11

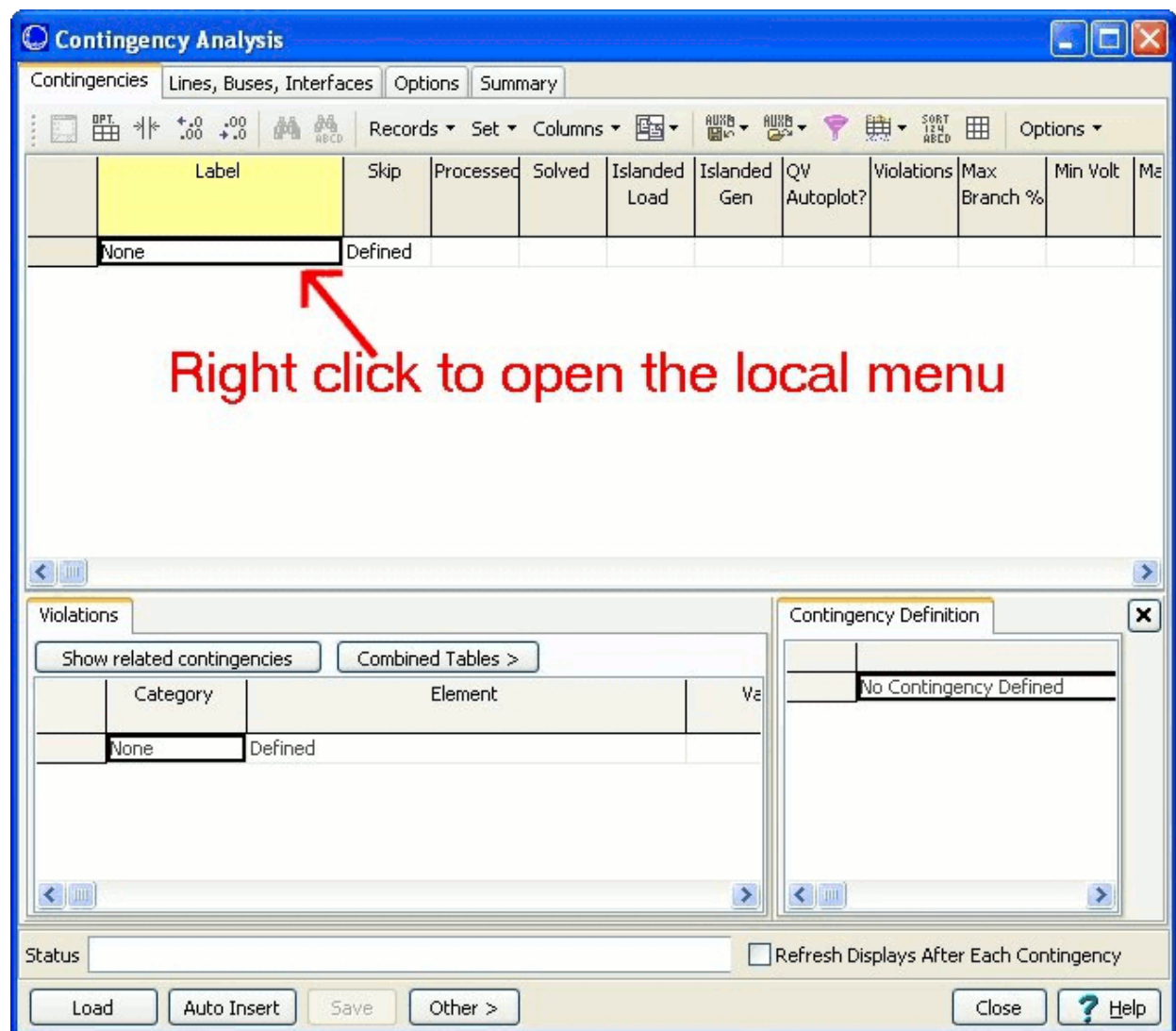
[Next](#)

Defining Contingencies

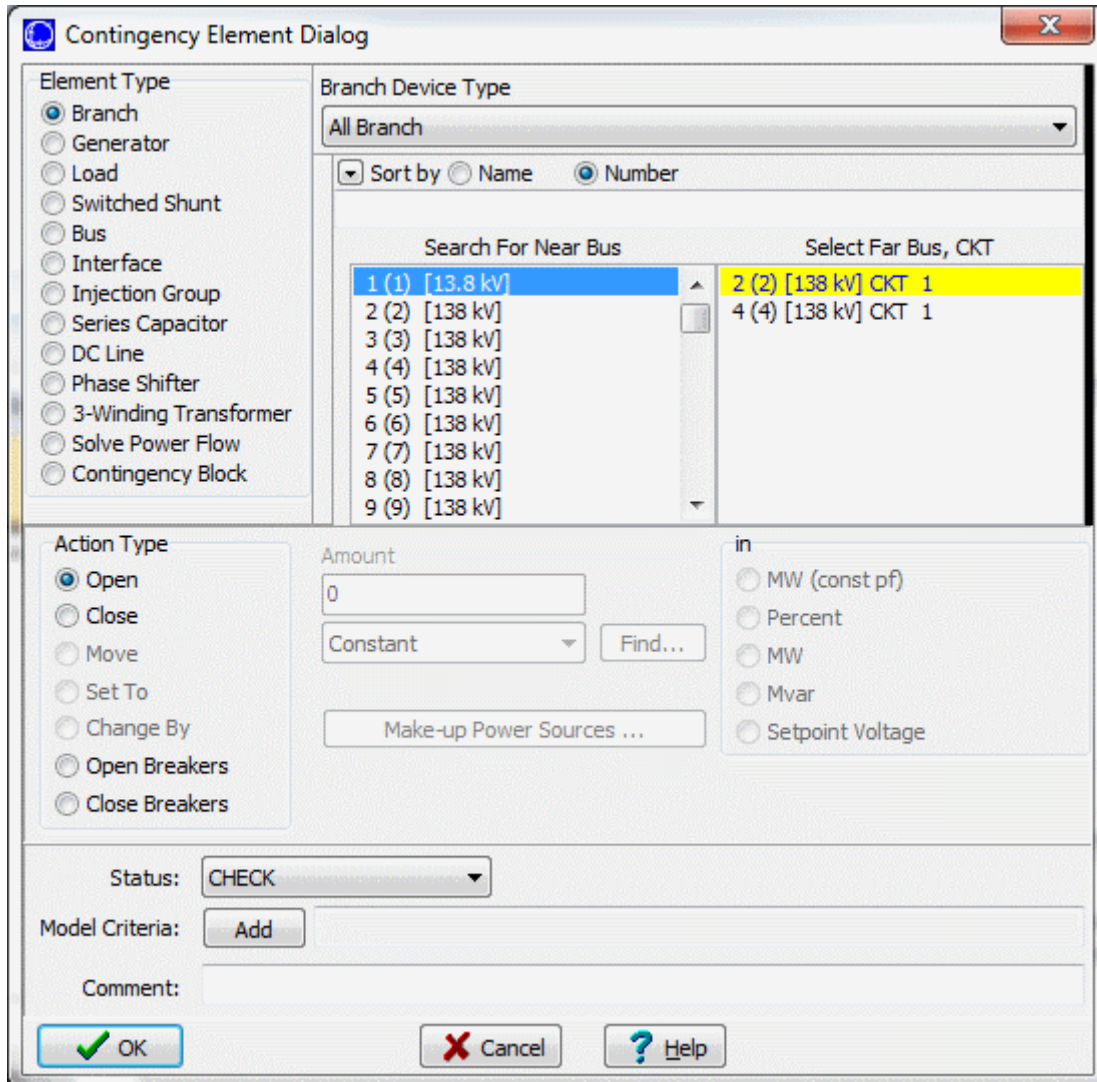
Contingencies can be either Single Element or Multiple Element. A single element contingency has only one associated contingent action. Click [here](#) for more information on the terminology used. There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert of Single Element Contingency**. This tutorial will utilize the Auto Insert and Quick Insert tools.

Quick Insert of Single Contingency Element

- Right-click on the grid (as shown below) and select **Insert Special > Quick Insert of Single Element Contingency** from the local menu. Simulator opens the [Contingency Element Dialog](#).



Contingency Analysis Dialog



Contingency Element Dialog

Note: The active fields in this dialog depend upon which **Element Type** is selected.

The defined action for this contingency will be to change the load at bus 2 by 25%.

- Select *Load* under **Element Type** and **2 (2) #1 [138 KV]** under **Choose the Element**.
- Select *Change By* under **Action Type**.
- Enter *25* for **Amount**, choose *Constant* from the drop-down box, then select *Percent* under the field labeled **in**.

Note that you can enter either positive or negative values in the Amount field to specify the direction that you want the change to occur (i.e. for a load contingency, entering a positive Amount increases the power consumed by the load, a negative value would decrease the load's power consumption).

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information. See Model Criteria and Comment in the [Contingency Element Dialog](#) for more information on those fields. For the purpose of this tutorial, we will not use these fields.

- Click **OK** to insert the contingency element.

The lower right portion of the Contingencies Tab contains a Contingency Definition list display. This display lists all elements (actions) associated with the selected contingency. Since we inserted a single element contingency, there is only one element shown in the Definition.

Contingency Definition				
	Actions	Model Criteria	Status	Comment
1	CHANGE LOAD AT BUS Two (2) BY 25.00 MW (cnst pf)		CHECK	

Contingencies Tab – Definition Display

The next portion of the tutorial provides an example of Auto Insertion of Contingency Elements.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, right-click in the Contingencies grid on the [Contingency Analysis Dialog](#), and select **Insert Special > Auto Insert Contingencies** from the popup menu. This opens the [Auto Insertion of Contingencies Dialog](#).

Auto Insertion of Contingencies Dialog

We will insert contingencies for all branches and generators. This will require two executions of the auto insert tool.

- Verify that **Single Transmission Line or Transformer** is selected.

When using the Auto Insert tool, you can limit the contingencies inserted to only those meeting a defined filter. We want to insert contingencies for all branches and generators so no filtering is desired.

- Click to remove the checkmark in **Use Area/Zone Filters**.
- Verify that no other options are selected.
- Click the **Do Insert Contingencies** button to accept the remaining default values and automatically insert the branch contingencies.
- Click **Yes** when asked to confirm the insertion of 11 contingencies.

Note that the Contingencies Tab of the Contingency Analysis Dialog now shows 12 contingencies.

- Right-click on the list display of the contingencies tab and select **Insert Special > Auto Insert Contingencies** from the local menu.
- Select **Single Generating Unit** then click the **Do Insert Contingencies** button. Click **Yes** to complete the auto insert of 4 generator contingencies.

Note that the Auto Insert tool did not insert a contingency for the generator connected to the Slack Bus. You can manually insert slack bus generator contingencies.

The contingencies tab now shows 16 contingency records. You can click on an individual record and view its information in the [Contingency Definition](#) section of the Contingencies Tab.

The next section of the tutorial discusses running the contingency analysis.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 4

When running the Contingency Analysis, the user has three options:

1. Run every contingency on the list of contingencies (click **Start Run** on the Contingencies Tab, click **Start** on the Summary Tab or select **Run Contingency Analysis** from the local menu of the list of contingencies)
2. Run a single contingency (discussed in the advanced tutorial section) or
3. Run a single contingency then save the post-contingency state as the new reference state (also discussed in the advanced tutorial section)

We will run every contingency in the list for this portion of the tutorial.

- Press Start Run on the Contingencies Tab. Note: **Pause** and **Abort** buttons are available on the dialog while the contingency analysis is running. These may prove useful when processing a long list of contingencies.

The results from the run are shown in the Contingency Analysis Dialog.

Contingency Analysis Dialog - Contingencies Tab

	Label	Skip	Processed	Solved	Islanded Load	Islanded Gen	QV Autoplot?	Violations	Max Branch	Min Volt	Max Volt	Max Interface %
1	L_00001One-00002TwoC1	NO	YES	YES			NO	1	149.3			
2	L_00007Seven-00005FiveC1	NO	YES	YES			NO	1	113.5			
3	L_00002Two-00006SixC1	NO	YES	YES			NO	1	103.9			
4	L_00002Two-00003ThreeC1	NO	YES	YES			NO	0				
5	L_00002Two-00005FiveC1	NO	YES	YES			NO	0				
6	L_00003Three-00004FourC1	NO	YES	YES			NO	0				
7	L_00004Four-00005FiveC1	NO	YES	YES			NO	0				
8	L_00002Two-00004FourC1	NO	YES	YES			NO	0				
9	L_00006Six-00007SevenC1	NO	YES	YES			NO	0				
10	L_00006Six-00007SevenC2	NO	YES	YES			NO	0				
11	L_00001One-00003ThreeC1	NO	YES	YES			NO	0				

Violations

Category	Element	Val
1 Branch Amp	One (1) -> Three (3) CKT 1 at One	406.00

Contingency Definition

Contingency	Definition
1	OPEN Branch One_138.0 (1) TO Tw

Status: Finished with 3 Violations and 0 Unsolveable Contingencies. Initial State Restored.

Buttons: Load, Auto Insert, Save, Other >, Start Run, Close, Help

Contingency Analysis Dialog – Contingencies Tab


Note: The **Refresh Displays after Each Contingency** option (lower right) can slow down the analysis significantly when running a long list of contingencies.

The contingency analysis results are sorted on the contingencies tab in descending order by worst violation. We see that there were three violations resulting from the contingency analysis (one for each of the first three contingency records displayed) and that no unsolvable load flows resulted (as shown in the Status Section of the Contingencies Tab.) If you enlarge the Contingency Analysis Dialog (by dragging either side or the corner of the pane) you see more information in the Status Window, specifically, "Finished with 3 Violations and 0 Unsolvable Contingencies. Initial State Restored."

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to: buses, switched shunts, limit groups, loads, branches, generators, areas / super areas, and power flow solution options.

Both prior to and following completion of solving a list of contingencies, the reference state is loaded into memory. This ensures that all contingency analysis solutions start from a common base case and that the system is restored to its initial state following a solution. The last sentence in the Status portion of the dialog, "Initial State Restored," tells the user that the simulation case was restored to the reference state upon completion of the Contingency Analysis Run. See [Contingency Case References](#) for more information on the Reference State.

The [Violations Section](#) (lower left) of the Contingencies Tab provides a description of each violation resulting from the execution of the contingency selected in the list of contingencies. Scroll through the list of contingencies to view information about the resulting violations for each.

Note: You can hide the Contingency Definition section of the Contingencies Tab by clicking the  button.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 5

The [Summary Tab](#) of the Contingency Analysis Dialog provides the status of the present contingency analysis run.

Contingency Analysis

Contingencies | Lines, Buses, Interfaces | Options | **Summary**

Summary

Applied:
OPEN Branch 2_138.0 (2) TO 6_138.0 (6) CKT 1 | | CHECK | | Opened flow of 2.72 MVA
Contingency L_000022-000066C1 successfully solved.
Solving contingency L_000033-000044C1
Applied:
OPEN Branch 3_138.0 (3) TO 4_138.0 (4) CKT 1 | | CHECK | | Opened flow of 53.38 MVA
Contingency L_000033-000044C1 successfully solved.
Solving contingency L_000044-000055C1
Applied:
OPEN Branch 4_138.0 (4) TO 5_138.0 (5) CKT 1 | | CHECK | | Opened flow of 12.18 MVA
Contingency L_000044-000055C1 successfully solved.
Solving contingency L_000077-000055C1
Applied:
OPEN Branch 7_138.0 (7) TO 5_138.0 (5) CKT 1 | | CHECK | | Opened flow of 60.79 MVA
Contingency L_000077-000055C1 successfully solved.
Solving contingency L_000066-000077C1
Applied:
OPEN Branch 6_138.0 (6) TO 7_138.0 (7) CKT 1 | | CHECK | | Opened flow of 25.42 MVA
Contingency L_000066-000077C1 successfully solved.
Solving contingency L_000066-000077C2
Applied:
OPEN Branch 6_138.0 (6) TO 7_138.0 (7) CKT 2 | | CHECK | | Opened flow of 25.42 MVA
Contingency L_000066-000077C2 successfully solved.
Contingency Analysis finished at July 25, 2007 14:26:04

Total # of contingencies	11	Start Time	7/25/2007 2:26:04 PM
# Processed	11	End Time	7/25/2007 2:26:04 PM
# Unsolvable	0	Total Run Time	0.46 Seconds
# Violations	1	Avg. Time per Ctg	0.042 Seconds

Status: Finished with 1 Violations and 0 Unsolvable Contingencies. Initial State Restored. ☐ Refresh Displays After Each Contingency

Contingency Analysis Dialog – Summary Tab

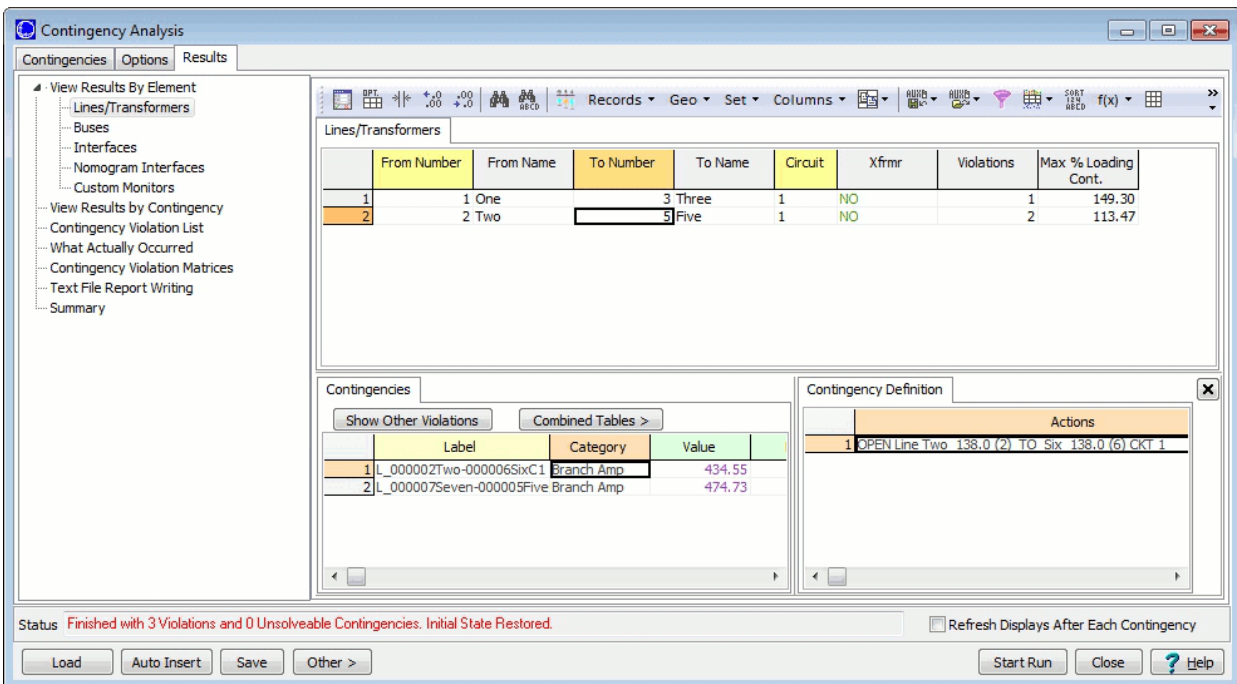
1 2 3 4 5 6 7 8 9 10 11

Prev Next

Tutorial: Contingency Analysis - Page 6

- Click on the View Results By Element page of the dialog.

The View Results By Element page contains five sub-pages: Lines/Transformers, Buses, Interfaces, Nomogram Interfaces, and Custom Monitors. The information contained on each of the sub-pages provides an alternate method of viewing information similar to that contained on the [Contingencies tab](#). The individual pages show model objects (subject to area/zone/owner filters) only if they are associated with specific contingencies.



Contingency Analysis Dialog – View Results By Element

With the exception of Custom Monitors, the user can select any model object on its respective sheet to see how many times a violation occurred on the device during the contingency analysis. When a particular device is selected, the [Contingencies](#) and [Contingency Definition](#) sections give the details of the specific contingencies that caused the violation (or violations) on the selected device. For example, on the Lines/Transformers page:

- Select the line from 2 to 5 as shown above. This line had one violation on the most recent contingency analysis run. The contingencies section shows which contingency caused the violation and the contingency definition section details the elements that define the contingency.
- Switch back to the Contingencies tab of the dialog.
- Scroll down in the list of contingencies and select the contingency labeled CHANGE LOAD AT BUS 2 (2)
- Right-click on the element displayed in the Contingency Definition Section of the Contingencies tab
- Select **Show Dialog** from the local menu.

- Modify the element's **Action** to: *Set To 600 MW (const pf)* using the **Action Type**, **Amount**, and **in** fields of the Contingency Element Dialog.
- Click **OK** to close the dialog. The Contingency Definition should now show SET LOAD AT BUS 2_138 (2) TO 600.00 MW (const pf).
- Click **Start Run** on the dialog and **Yes** when asked to confirm.
- Following the run, the Status field now shows, "Finished with 7 Violations and..." The increase in load resulted in four contingency violations not present during the last run.
- Again select the contingency labeled SET LOAD AT BUS 2... on the list display. The Violations section shows the four branch violations that occurred for the selected contingency.
- Now switch to the Lines/Transformers sub-page of the View Results By Element page.
- Select the line from 2 to 5. This line experienced violations under two contingencies during the run. The Contingencies section now shows the details of both contingencies (opening line 5 to 7 and changing the load at bus 2) that caused overloads on the line. Notice that the information in the Contingency Definition section is specific to the Contingency selected in the Contingencies section.

Note: The **Show Related Contingencies** and **Show Other Violations** buttons (on the Contingencies and View Results By Element pages respectively) provide a fast method of switching between the two pages and viewing related information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

You now have enough information to effectively utilize PowerWorld Simulator's Contingency Analysis tool. The remaining portions of the tutorial introduce the full capabilities of the Contingency Analysis tool and provide links to the applicable Help files.

Contingency Records

There are four methods of defining contingencies. See [Defining Contingencies](#) for more information. Contingency Records can also be [saved to](#) or [loaded from](#) a file.

Contingency Blocks and Global Actions

The user may desire to have a common set of actions occur during more than one (or all) contingencies. Instead of repeatedly defining the same contingency element (or elements) in multiple contingencies, Simulator provides the option to use Contingency Blocks and/or Global Actions.

A [Contingency Block](#) is a set of contingency actions that can be defined and then called upon by individual contingency records.

[Global Actions](#) are actions defined by the user that will occur during all contingencies unless an Inclusion Filter is specified that allows defining with which contingencies the action will be included.

Contingency Solution Options

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options defined for the contingency will be used. Other options set to *use default* will look to the Contingency Analysis Options. Again, any options defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *use default* will be set to the same setting as the power flow solution options.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Reference State

The [Reference State](#) (introduced on page 4 of the tutorial) can prove very useful to advanced users if they know how to exploit its capabilities. When running a list of contingencies the reference state is loaded into memory prior to executing each contingency. The system is restored to the reference state following the contingency analysis run. In addition to running an entire contingency list, the user also has the option to solve individual contingencies by right clicking on the desired contingency - on the Contingencies tab of the Contingency Analysis Dialog - and selecting either **Solve Selected Contingency** or **Solve and Set as Reference** from the local menu.

Solve Selected Contingency causes Simulator to first load the reference state into memory then solve the contingency. THE SYSTEM STATE IS NOT RESTORED TO THE REFERENCE STATE FOLLOWING THE SOLUTION; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to the reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to the reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

Solve and Set As Reference acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

Make-up Power Sources

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Contingency Elements

A contingency element consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

Contingency Action - Click [here](#) for a list of available contingency actions.

Model Criteria - Model Criteria are criteria under which the contingency action will occur and consist of both [Model Conditions](#) and [Model Filters](#).

Status – The status field of a contingency element can take one of four values: CHECK, ALWAYS, NEVER or POSTCHECK.

- CHECK - The action will be executed if the Model Criteria are true or if no Model Criteria are specified. CHECK is the default status setting.
- ALWAYS - The action will always be executed, regardless of the Model Criteria.
- NEVER - The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- POSTCHECK – Following completion of all CHECK and ALWAYS actions, the contingency analysis tool runs the load flow solution. POSTCHECK actions are then addressed recursively until all are complete. The execution of POSTCHECK actions follows the same rules as CHECK actions with the exception of not being checked until the load flow has been solved. If the Model Criteria specified for the POSTCHECK action are met in the solved load flow solution (or if no Model Criteria are specified), then the action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped.

For more information, see [Contingency Definition Display](#).

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Contingency Analysis Dialog – Options Tab Overview

The bulk of the [Options Tab](#) contains five groupings, each of which concerns a different aspect of the contingency analysis. A brief description of the actions allowed via each grouping follows. Click on the headings below for more information on each.

Modeling

The modeling group allows the user to:

- Define the Contingency Analysis Load Flow Calculation Method
- Specify Limit Monitoring Settings
- Instruct Simulator to retry the solution using the Robust Solution Process following a failure to converge
- Specify the use of specific solution options for contingencies
- Specify Make-up Power for the post-contingency solution
- Define Bus Load Throw-Over Records – BLTRs provide the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system during a contingency
- Define Generator Maximum MW Responses in Post-Contingency – allows the user to limit the absolute MW response of a generator during a contingency
- Define Generator Line Drop Compensation (LDC) and Reactive Current Compensation (RCC) – LDC and RCC controls allow the user to model the real-time control done at some real generators
- Specify a post-contingency aux file to be loaded at the start of each contingency. This allows the setting of very specialized post contingency settings.
- Specify how reactive power is modeled for DC calculation methods

Limit Monitoring

The Limit Monitoring grouping contains the following:

- [Advanced Limit Monitoring](#) page that allows you to shape how limit violations are detected and reported
- [Monitoring Exceptions](#) display

Contingency Definitions

The Contingency Definitions grouping allows the user to:

- Define Contingency Blocks and Global Actions (refer to [Page 7](#) of the tutorial)
- Define Model Criteria (Expressions, Conditions and Filters)
- Specify Model Criteria Options

[Distributed Computing](#)

This page contains special distributed computing options for contingency analysis.

[Miscellaneous](#)

The Miscellaneous sub-tab provides options pertaining to the loading and saving of contingency records as well as specifying how the reference state should be established when the contingency analysis tool is accessed.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

Contingency Analysis Dialog – Other Contingency Actions

The **Other >** button on the [Contingencies tab](#) and the [View Results By Element page](#) of the [Contingency Analysis dialog](#), provides access to a number of additional contingency actions. Some of the actions available include:

- Deleting all contingencies
- Clearing contingency results
- Setting or restoring the reference state
- Producing combined tables of results
- Producing detailed reports of results
- Comparing lists of contingencies
- Filtering results
- Auto-filling blank comment fields and performing sensitivity calculations such as OTDFs and PTDFs.

For more information, see [Other Contingency Actions](#).

1 2 3 4 5 6 7 8 9 10 11

Prev

Time Step Simulation

The Time Step Simulation tool allows you to specify operating conditions and obtain power flow solutions for a set of points in time. It provides the tools needed to analyze the operation of a power system hour by hour or by intervals down to one second.

Time Step Simulation is available in the base Simulator package. If you own Simulator, you can start taking advantage of this valuable tool right away. In addition, if you own Simulator OPF or SCOPF licenses, you can solve OPF and SCOPF scenarios on a time point to time point basis and use the tool to evaluate the behavior of prices and operating constraints. The tool will obtain the optimized generation dispatch for each time point of the analysis horizon.

In order to access the Time Step Simulation, go to the [Tools](#) ribbon tab, and choose **Time Step Simulation** from the [Run Mode](#) ribbon group. This is only available in [Run mode](#). The following topics describe key components of the time step simulation tool:

- [Time Step Simulation Quick Start](#)
- [Time Step Simulation Dialog](#)
- [Specifying and Maintaining a List of Time Points](#)
- [Loading Input Data](#)
- [Setting up Scheduled Input Data](#)
- [Storing Input Data and Results](#)
- [Running a Timed Simulation](#)
- [Time Step Simulation Toolbar](#)
- [Running OPF and SCOPF Simulations](#)
- [Application of Controller Time Delays](#)

Time Step Simulation: Quick Start

This topic will get you started with using the [Time Step Simulation tool](#) and help you become familiar with the basics of setting up a time step simulation run.

The [Time Step Simulation dialog](#) is displayed when you access the Time Step Simulation by selecting **Time Step Simulation** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. This dialog contains several pages. The **Summary** page is used to define and control the time points you want to analyze. As an example, we will assume that you have hourly load data for tomorrow and that you want to determine the system bus voltages for each hour. You would do the following:

Step One: Set the List of Time Points

In order to create a list of points, right-click on the **Summary** page and select **Insert New Timepoint(s)**, which brings up the [New Timepoint Dialog](#). In this dialog, select tomorrow's date from the drop down calendar component. Set the field **Total Number of Timepoints to Enter** to 24. Assume the other default values are acceptable and click OK. This will insert 24 time points, one for each hour starting tomorrow at 12:00 AM.

Step Two: Specify Input Data

Input data is specified on the **Input Page**. In this example your input data corresponds to hourly loads. Select the **MW Loads** sub-page. In order to specify hourly load values you have to insert a column for each load. Right-click on the grid and select **Time Point Records > Insert/Scale Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In the selector component, select the load for which you want to specify hourly values. You can press the shift key to select multiple elements. Then press the blue **Arrow Button** to pass the selected loads to the right side of the selector. Select **OK** to insert the new column(s). Now you can specify the hourly MW values for those loads.

Step Three: Specify the Custom Results

Obtaining solutions for a large number of time points has the potential to create unnecessary burden in memory and storage due to the large amount of data that can be generated. For this reason, the Time Step Simulation tool allows you to explicitly specify what quantities you want to display and store. This is done on the [Results page](#). This page contains grids for many devices including Generators, Lines, etc. In this example we want to analyze the bus voltage magnitudes. We specify what quantities we want to store as results for each object by clicking the **View/Modify** button, which brings up the [Custom Results Selection Dialog](#). In this dialog select the **Buses** page and click the **Add/Remove Fields** button to bring up the list of **Available Bus Fields**. Find the **Per Unit Voltage** field and **Add** this to the list of **Selected Fields** and then click OK when done modifying fields. Results will be saved only for those buses that have the **Time Selected** field on the grid set to YES. Set this field to YES for the buses that you want to save. Click the **Save and Close** button to save the custom results settings, i.e., which objects, fields and records are to be kept during the solution; in this case bus voltages.

Step Four: Run the Simulation

The upper part of the Time Step Simulation dialog contains buttons used to control the simulation. To do a full run of the 24 hours click the **Do Run** button. The **Last Result** box shows the progress of the simulation as each time point is being solved. If you are in the **Results – Buses** page, you will see that a column was added for each bus set to YES in the **Custom Results Selection Dialog**. Each column shows the bus per unit voltage. Recall that you can right-click on any Simulator grid and select **Plot Column** to obtain a plot of the column values. If you select cells spanning all the bus per unit voltage columns, you will obtain the voltage profiles for each bus, versus time.

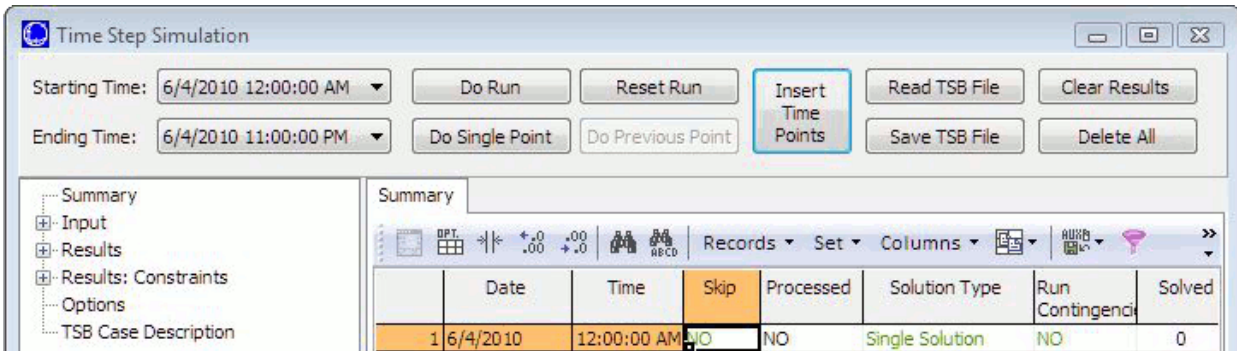
If you are in the **Summary** page, the **Processed** column shows that each point was in fact processed and solved.

Step Five: Save the Results

Once you have completed the simulation, you can save the results in a **Time Series Binary** (.tsb) file by pressing the **Save TSB File** button. This file will contain all the input data, the simulation options, the custom results settings, and the results. You can reload this .tsb file any time by pressing the **Read TSB File** button.

Time Step Simulation Dialog

The Time Step Simulation dialog is used to control and visualize the time simulation. The top section of the form contains buttons for data input/output and buttons to control the progress of the simulation. The main part of the form has a number of [Time Step Simulation pages](#) that contain grids and options where input data can be specified and simulation results can be examined.



Input/Output Buttons

Insert Time Points

Press this button to open the [New Timepoint Dialog](#) that allows specification of time point starting date and time, number of time points to enter, and the interval between time points.

Read TSB File

Press this button to read a Time Series Binary file (.tsb File). This file stores input and scheduled data, the simulation options and the results.

Save TSB File

Press this button to save the input, custom inputs and scheduled data, the simulation options, and the results and custom results in a Time Series Binary file (.tsb file).

Clear Results

Press this button to clear any results stored with the time step simulation.

Delete All

Press this button to delete all information associated with a time step simulation including, results, selection of objects and fields for results, schedules, schedule subscriptions, and time points.

Simulation Control Buttons

During the solution, the Time Step Simulation solves each time point in a sequential manner. During this process, the Simulation can be in one of three states:

1. **Reset:** When the simulation has not started, when it has been completed, or when it has been paused and then reset.
2. **Running:** When the simulation is solving time points sequentially.

3. **Paused:** When the user has paused the simulation. The simulation actually waits until the present time point is solved in order to pause. If the simulation includes an SCOPF solution, all the contingencies are processed and the system is optimized before the simulation is paused. Note that once a time point is solved its results are available on the various grids.

The user controls the Time Step Simulation by means of the following control buttons:

Starting Time

Use this selector to specify a start time point other than the first time point in the list. The simulation will disregard the time points before the selected start time point.

Ending Time

Use this selector to specify an end time point other than the end time point in the list. The simulation will stop after processing the selected end time point.

Do Run [Pause Solution, Continue Solution]

Press this button to initialize the Time Step Simulation and go through all the time points until the last time point or a time point with the **Skip** field set to *Pause* is found. As the simulation takes place, the **Last Result Box** will be updated with messages. In addition, the [Summary page](#) will change the **Processed** and **Solved** fields from NO to YES, reflecting the simulation progress. The [Results](#) grids will be updated once the solution for a time point has been found.

When the Simulation starts, the simulation status changes to **Running**, and the **Do Run** button changes its caption to **Pause Solution**. Press this button to pause the Solution. The simulation will continue until the current time point is solved entirely. Once the Simulation status is set to **Paused**, the **Do Run** button caption changes to **Continue Solution**. Press this button to continue the solution.

Do Single Point

Use this button to solve the next time point. You can see the last processed point in the **Last Result** box. You can also select a specific starting point by using the **Starting Time** selector.

Do Previous Point

Use this button to solve the previous time point. You can see the last processed point in the **Last Result** box. You can also select a specific starting point by using the **Starting Time** selector.

Reset Run

Use this button to go back to the first time point and initialize the simulation. This action does not delete the results of the time points processed so far, but it resets all the **Processed** fields to NO.



The screenshot shows a horizontal control bar. On the left, there is a label 'Last Result:' followed by a text box containing the word 'Initialized'. To the right of this is a 'Present Time' label followed by an empty text box. On the far right, there is a checkbox labeled 'Refresh All Displays Each Time Step'.

Last Result

The last result box in the lower left corner of the dialog is used to show solution progress messages. The messages indicate correct solutions or errors in the solution of the particular time point. It also shows the progress of the contingency analysis during a SCOPF solution.

Present Time

If you are solving the time points using a [timed simulation](#), the Present Time of the process is displayed in this field.

Refresh All Displays Each Time Step

Check this box to have all displays refresh after each time step. This could slow the process down if there are a number of displays open.

Time Step Simulation Toolbar

The purpose of this toolbar (besides providing shortcuts) is to command a [Time Step Simulation](#) without having to keep the [Time Step Simulation dialog](#) in a non-minimized state. This is particularly important during a [Timed Simulation](#), in which you want to see the changes on the oneline diagram as they occur in time.



The figure shows the main functions of the toolbar. [Reset/Solve Previous Point/Play/Solve Next Time Point/Pause Control Buttons; Last Result Box; Present Time; Progress Bar, and Timed Simulation Options]. All of the buttons are disabled until a list of time points have been defined in the [Summary page](#). The Time Step Simulation Toolbar is available in Run mode and is visible only when the [Time Step Simulation dialog](#) is open. Minimize the Time Step Simulation dialog to prevent it from obscuring the oneline.

Time Step Simulation Pages

The [Time Step Simulation dialog](#) contains a number of pages used to specify input and scheduled data, and to examine the results of the simulation. See the following topics for a detailed explanation of each page:

[Summary](#)

[Input](#)

[Results](#)

[Results: Constraints](#)

[Options](#)

[TSB Description](#)

Time Step Simulation: Summary

The Summary page of the [Time Step Simulation dialog](#) is used to define the time points and display a summary of the simulation. Most of the commands to manage time points can be accessed from the [Summary page local menu](#).

Once time points are defined, the Summary page presents the following columns for each time point:

Date

The date of the time point. To modify the date, right-click and select **Time Point records > Change Timepoint Time** on the local menu. This will open the [Change Timepoint Time Dialog](#).

Time

The time of the time point to the second. To modify the time, right-click and select **Time Point records > Change Timepoint Time** from the local menu. This will open the [Change Timepoint Time Dialog](#).

Skip

Set this field to YES to include the time point in the simulation. If set to NO, the input data and any scheduled actions that occur at this time point are not applied to the power system.

Processed

This field is set to YES if the simulation has processed the time point. This includes applying the input data, applying the scheduled actions, and solving the power flow for that particular time point.

Solution Type

The time point can be solved using one of the following **Solution Types**: Single Solution, Unconstrained OPF, optimal power flow (OPF), and security-constrained optimal power flow (SCOPF). The last three solution types require the Simulator OPF/SCOPF add-ons. See [Running OPF and SCOPF Time Step Simulations](#) for more information.

Run Contingencies

If the case has a defined set of [contingencies](#), you can choose to run the contingency analysis for each time point by setting this option to YES.

Solved

This will be set to an integer value indicating at what point a solution was obtained for the time point under the specified Solution Type.

Num Loads

Total number of loads for which input data has been specified.

Total MW Load

Total MW load specified as input data for the time point.

Total Mvar Load

Total reactive load specified as input data for the time point.

Num Gens

Total number of generators in the system for which input data has been specified.

Total MW Gen

Total MW of generation specified in the input data.

Pre Script Cmd

The Time Step Simulation has the capability of running a pre script command before each time point is solved. The pre script command can be run either before applying the input data and scheduled actions or right after applying the input data and scheduled actions but is always run before the power flow solution is solved. See [Time Step Simulation: Options](#) for more information on specifying when the pre script command is run. See the [Script Command](#) section to learn the details about performing Simulator actions using script commands.

Post Script Cmd

The Time Step Simulation has the capability of running a post script command right after the time point is solved. The post script command can be run either before storing results or after storing results. See [Time Step Simulation: Options](#) for more information on specifying when the post script command is run. See the [Script Command](#) section to learn the details about performing Simulator actions using the script language.

Time Step Simulation: Summary Local Menu

The local menu of the [Time Step Simulation Summary page](#) is used to perform a number of logical actions on the time point grid and is, as every Simulator local menu, accessed by right-clicking anywhere on the grid. When selected from the Summary page, the local menu shows the following additional time step simulation specific options:

Time Point records > Apply Time Point

Simulator applies the input data of that particular time point to the power system model, without solving the time point. Upon selection of this option, the time point information can be visualized in the [Case Information Displays](#).

Time Point records > Solve Time Point

This option applies the time point input and the scheduled data to the power system and solves the time point with the specified Solution Type.

Time Point records > Change Timepoint Time

This option allows the user to modify the date/time of the selected time point through the [Change Timepoint Time Dialog](#). If the new date/time belongs to an existing point in the list, an error message is displayed. The time points can be specified with a precision of up to 1 second.

Insert New Timepoint(s)

Use this option to insert new time points into the time point list. This selection brings up the [New Timepoint Dialog](#).

Delete Entire Timepoint Record

Use this option to delete the selected time point and all of the input data and custom results for that time point.

Set/Toggle/Columns > Plot Column(s)

Use this option to automatically generate column plots. By default, in the Time Step Simulation grids, the column plots display graphs of column data versus the date/time column (the combination of the date and time columns).

Time Step Simulation: Input

The [Time Step Simulation dialog](#) Input page is used to specify input data per time point, scheduled data, time step actions, and custom inputs. This requires that the list of time points have been created. See the [Specifying and Maintaining a List of Timepoints](#) topic for details on managing the list of time points.

The Input page contains several pages that can be grouped in four types: Input Pages, Schedule Pages, Time Step Actions Page, and Custom Inputs Pages.

Input Pages

The time point input pages (MW Loads, Mvar Loads, Actual MW Generation, Maximum Generation, Line Status, Area Loads, Zone Loads, and Injection Groups) are [Matrix Grids](#) that are used to specify data on a time point by time point manner. In order to tell Simulator that we want to specify time point data for a particular object type (e.g. load, generator, line, etc.) we need to add that particular object to the corresponding grid. For instance, suppose that you want to specify MW data for Load 1 at bus 1. Go to the MW Loads page, right-click and select **Time Point records > Insert/Scale Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In this dialog, you can select the load and add it as a column to the MW Loads grid. The corresponding values at each time point for that particular load can then be entered in the grid.

Use the Area Loads and Zone Loads pages to specify the MW load for an entire Area or Zone at a particular time point. When Simulator applies the input data to solve a time point, it will scale the load of the specified area or zone to match the value entered for that time point.

Use the Injection Groups page to specify the total MW injection from an injection group at a particular time point. All loads and generators in an injection group will have their MW injection changed in proportion to their participation factor specified with the injection group to meet the specified MW injection. Generators will have their minimum and maximum MW limits enforced. Mvar injections will not be modified as part of this input. If a zero injection value is specified for a time point, no modifications will be made to the injection group for that time point. To zero out the injection, change the value to a very small number, e.g. 0.01.

Schedule Pages

The schedule pages ([Schedules page](#) and [Sched Subscriptions page](#)) are used to specify scheduled input data, i.e., data that more naturally spans multiple time points rather than being defined at each time point. Examples of data that can be scheduled are scheduled transactions, generator statuses, line statuses, etc. For a detailed explanation on how to set up scheduled data, please read the [Setting Up Scheduled Input Data](#) section.

Time Step Actions Page

Use this page to specify [Time Step Actions](#). These are conditional actions that can be applied during a time step simulation run if their Model Criteria evaluates to true for a specified time period.

Custom Inputs Pages

Custom inputs allow the selection of specific objects, object numeric fields, and the specification of what each selected field should be set to for a particular time point.

The top portion of this page is used to select the objects and fields and specify how the selection of objects and fields should be displayed in the grids contained on the bottom portion of the page. The selection of objects and fields for custom inputs is identical to the selection of objects and fields on the [Results page](#).

Once the objects and fields have been selected, the grids will be updated with a column for each of the selected object and field combinations. Use these columns to specify the values for the selected fields at each time point.

Matrix Grids

Matrix Grids are a special type of data grids used in the input and results pages of the Time Step Simulation tool. In these grids, the time dimension is assigned to the rows and the object fields (load MW, generator MW output, etc) are assigned to the columns. The column header corresponds to the ID of the object. The number of columns of the grid depends on the user options and simulation results:

Hourly Input Pages

In the case of input pages, the user has to specify hourly data for each object, e.g., load, generator, area. Thus the user adds each column to the grid explicitly.

Custom Result Pages

In the custom result pages, a column is created for each object whose results have been specified to be stored.

Constraint Pages

Since the constraints are determined at solution time, the columns appear only when constraints have been detected during the solution.

Time Step Simulation: Results Constraints

The Results Constraints page of the [Time Step Simulation dialog](#) is used to convey results that are specific to an optimal power flow type of study. This page is only available with the [OPF](#) and [SCOPF](#) add-ons of Simulator. When the OPF solution type is used to solve a time point, the status of the system during that particular time is optimized so that the total operating cost of the system is minimized, and the normal operation constraints are enforced. In addition, the SCOPF solution type enforces contingency constraints. The OPF and SCOPF solutions contain information about the elements that determine the LMPs, the binding constraints, the violating contingencies, and changes in the control settings.

The information related to contingencies on these pages is available only in the SCOPF Simulator add-on.

The pages of the Results: Constraints page are:

Results Summary

This page is similar to the [Summary Page](#), except that it contains additional operating information such as:

Initial Cost

The cost of operating the system given the initial generator set points and a standard power flow solution. No controls are moved to minimize cost.

Unconstrained Cost

The total operating cost after the controls are moved to minimize cost, without enforcing any normal operation or contingency constraint. The result is an operating cost that corresponds to an Economic Dispatch solution.

Unconstrained LMP

The average marginal price of the system under unconstrained optimization.

Final Cost

The total operating cost after a constrained (OPF or SCOPF) solution has been obtained.

LMP

(Average, Standard Deviation, Minimum and Maximum): Metrics of the LMP values.

Binding Lines

Number of transmission lines and transformers that are binding after the OPF/SCOPF solution has been determined.

CTGs Unsolvable

Number of unsolvable contingencies. These are severe contingencies that would cause the power flow solution to fail for that particular time point scenario. Unsolvability of the power flow case is related to maximum loadability conditions.

You can access the details of the binding elements by right-clicking and selecting [Show Binding Constraint Dialog](#) on the local menu.

Binding Lines

This is a [matrix grid](#) that shows information for transmission lines and transformers that become binding constraints during the OPF or SCOPF solutions. A similar page is available for binding interfaces.

When transmission lines or transformer thermal ratings become binding constraints for a time point, a column is automatically added, forming in this manner the matrix grid. You can access the details of the binding line or transformer by right-clicking and selecting [Show Binding Constraint Dialog](#) on the local menu. The grid also shows the following summary columns:

Processed

Gets set to YES if the time point was correctly processed.

CTGs with Viols

Number of contingencies that presented one or more violations (violating contingencies).

CTGs Unsolveable

Number of unsolvable contingencies for the time point.

BC Line Viols

Number of transmission line and transformer violations that were identified in the base case.

Line Viol

Number of transmission line and transformer thermal violations.

Binding Lines

Number of binding lines in the OPF/SCOPF solution.

Line Unenforceable

Number of lines with unenforceable limits.

Binding Interfaces

This is a [matrix grid](#) that shows information for interfaces that become binding constraints during the OPF or SCOPF solutions. A similar page for lines is available.

When interfaces become binding constraints for a time point, a column is automatically added to the matrix grid. You can access the details of the binding interface by right clicking and selecting [Show Binding Constraint Dialog](#) on the local menu. The grid also shows the following summary columns:

Processed

Gets set to YES if the time point was correctly processed.

CTGs with Viols

Number of contingencies that presented violations (violating contingencies)

CTGs Unsolveable

Number of unsolvable contingencies for the time point.

BC Interface Viols

Number of interface violations that were identified in the base case.

Interface Viol

Number of interface violations.

Binding Interface

Number of binding interfaces in the OPF/SCOPF solution.

Interface Unenforceable

Number of interfaces with unenforceable limits.

Binding Contingencies

This [matrix grid](#) shows information similar to that found on the Binding Lines and Binding Interfaces pages. Here though the data is organized by contingencies, which allows easy identification of the most severe contingencies. Each column of the matrix grid corresponds to a contingency. You can access the details of the binding element by right-clicking and selecting [Show Binding Constraint Dialog](#) on the local menu. The grid also shows the following summary columns:

Processed

Gets set to YES if the time point was correctly processed.

CTGs with Viols

Number of contingencies that presented violations (violating contingencies) .

Line Viol

Number of transmission line and transformer thermal violations.

Binding Lines

Number of binding lines in the OPF/SCOPF solution.

Binding Interfaces

Number of binding interfaces in the OPF/SCOPF solution.

Binding Line Summary Matrix

This [matrix grid](#) shows the number of time points for which a line has been binding due to each contingency. In this case, the rows correspond to binding lines, and the columns to violating contingencies. The grid also shows the following summary columns:

From Number

Binding line from bus number

To Number

Binding line to bus number

Circuit

Binding line circuit ID

Total Hrs

Total number of time points for which the line was binding (in the overall simulation).

Total Hrs Unenforceable

Total number of time points for which the line constraint was unenforceable (in the overall simulation).

Binding Line Summary List

Shows information similar to the **Binding Line Summary Matrix** but in form of a list. The same line may appear several times under different contingencies. Besides the information on the **Binding Line Summary Matrix**, this grid shows the following fields:

Contingency Name

Contingency that causes the line constraint to be binding

Avg MC

Time Average MVA Marginal Cost

Max MC

Time Maximum MVA Marginal Cost

Min MC

Time Minimum MVA Marginal Cost

Time Step Simulation: Binding Elements Dialog

This dialog is called from the [Results: Constraints Page](#) of the [Time Step Simulation dialog](#). Right-click on a time point record in any of the Results Summary, Binding Lines, Binding Interfaces, or Binding Contingencies tables, and select **Show Binding Constraint Dialog** from the local menu. The dialog shows the binding line and interface constraints determined by the OPF/SCOPF solution. The **Time** selector allows easy navigation through the list of time points.

The grid section of the dialog shows the following:

Type

Either line or interface flow

Constraint ID

Line or Interface ID

Contingency Name

Name of the contingency under which the constraint becomes binding.

MVA Marg. Cost.

Marginal cost of enforcing the constraint. For lines, the limit corresponds to the thermal (MVA) limit. For interfaces, it is given by a MW limit.

Time Step Simulation: Results

The results of the [Time Step Simulation](#) are presented on the grids of the Results page. The Time Step Simulation allows you to specify what objects (buses, lines, generators, loads, etc.) and what object fields (bus voltage, bus LMP, gen MW, etc.) should be displayed on the results grids. This gives the user the flexibility needed to explore the relevant results, avoiding at the same time the problem of storing a massive amount of results, most of which may not be relevant. Storage is a critical aspect of the Time Step Simulation, since a set of results comparable to full a PF/OPF/SCOPF solution is generated for each timepoint.

The Results page has two sections: The top section is used to set up the options needed to customize the results display. The grid section is used to display the actual results.

Results Page: Top Section

This section includes the following options:

View/Modify

Press this button to access the [Custom Results Selection Dialog](#). This dialog is used to specify the objects and object fields for which results will be stored.

Load

The custom result definitions set up in the [Custom Results Selection Dialog](#) can be saved in a Custom Results File (*.tsc) in order to use them with different *.tsb files or Simulator cases. Press this button to Load the results file and apply the result definitions to the current time step simulation.

Save

Press this button to save the result definitions in a Custom Results File (*.tsc).

Group Results by

The results grids for Areas, Buses, etc. are [Matrix Grids](#) that present the time point results for each type of object. For instance, suppose that we want to store the bus voltage magnitude and angle. The Buses grid will show columns for the time point values of voltage magnitude and voltage angle of selected each bus.

Objects: The columns will be grouped by objects, e.g., all the fields of bus 1, then all the fields of bus 2, etc.

Fields: The columns will be grouped by fields, e.g. all the bus p.u. voltages, then all the bus LMPs, etc.

Identify Results by

The results grids for Areas, Buses, etc. create one column for each object field specified in the [Custom Results Selection Dialog](#). The column header thus identifies the particular object for which time point results are displayed. This identification can be made based on the object **Number**, **Name** or **Number + Name** combination.

Results Page: Grid Section

This grid section is used to display the results of the [Results Grids](#).

Time Step Simulation: Custom Results Selection Dialog

This dialog is used to specify what objects and object fields will be stored during a [Time Step Simulation](#) and will be displayed on the [Results grids](#). This dialog is opened using the **View/Modify** button on the [Results](#) page of the [Time Step Simulation dialog](#).

To specify that a particular field be stored for a certain object:

1. Click on the page of the object type to store (Areas, Buses, etc.)
2. Set to **YES** the **Time Selected** field of those objects for which you want to store information.
3. Click the **Add/Remove Fields** button to open the Select Fields dialog. Fields contained in the Selected Fields list will be stored for each of the selected objects. To move fields between the Available Fields and Selected Fields lists, drag the fields between lists or select fields and use the Add and Remove buttons as appropriate. Once the fields are selected click the OK button to save the changes and close the dialog.

Once you are done with the selections, press the **Save and Close** button to apply the customization. The [Results grid pages](#) will be filled with the corresponding columns after the Time Step Simulation starts.

Note that the actual results and the result customization will be stored in the .tsb file. For more information about saving the Time Step Simulation results, please read the [Storing Input Data and Results](#) section.

Fields for Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Loads, Owners, Super Areas, Switched Shunts, Transformers, and Zones can be specified to be displayed on the grids and stored in the .tsb file.

Time Step Simulation: Results Grids

These are the grids of the [Results page](#) that contain [Time Step Simulation](#) results based on the fields and objects selected using the [Custom Results Selection Dialog](#). The fields and objects must be selected before the time step simulation run is started in order for those results to show up in the grids.

There is one page for each type of object: Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Loads, Owners, Superareas, Switched Shunts, Transformers and Zones.

These pages are all [Matrix Grids](#). Each column of a grid corresponds to a field of a specific object of the power system, e.g., KV voltage of bus 1. Each row of the grid corresponds to a timepoint.

Note that you can plot the results of the grid versus the timepoint date time by right-clicking on the grid and selecting **Set/Toggle/Columns > Plot Column(s)** from the local menu.

Time Step Simulation: Options

The Options page of the [Time Step Simulation dialog](#) contains options to control how input data is applied and how output results are presented and stored.

Area and Zone Load Scaling

In many practical studies, data of individual MW loads may not be available for all the loads in a control area for each time point. In order to simulate load variations, Simulator allows you to specify the Area or Zone Total MW Load values for each time point in the **Area Loads** and **Zone Loads** pages of the [Input page](#). Thus, the Time Step Simulation load data may be a combination of:

- Areas or zones where each individual load is specified
- Areas or zones where only the total MW load is known
- Areas or zones where some individual loads and the total area load are known

If you have specified individual load values for a load within an area or zone that is also set to be scaled to a total MW value, Simulator will adjust the individual load to the individual value first, and then the load will be scaled with the rest of the loads in the area or zone to achieve the total MW value desired for the time point. The final load will be the total value specified for the area or zone.

When time point values are specified for areas and zones within the same time step input, only one or the other can be used. This can be set in the **Use Time Point Values of** option.

You can also specify in the **Reactive Power Scaling** option how the reactive power is treated when the load is scaled. You can choose to have the reactive power scaled to keep the power factor constant, or for the reactive power to remain fixed at its original value.

Solution Options

These options control the solution process.

Pause if Power Flow Does Not Solve

The simulation is stopped at the time point where a power flow solution cannot be obtained. This is an indication of wrong data or a system brought to its loadability or transfer capability limit.

Enable Power Flow Area Interchange Control

This option ensures that the area interchange control, if possible, is enforced in the case.

Turn generators off AGC when output changes

At each time point, generators can be adjusted by specifying Gen Actual MW values on the Input page or as part of Injection Groups that have their injection changed on the Input page. If area interchange is enabled for the case and associated area or super area where generation changes have been made, this may cause the generators to move away from their values specified on the Input pages. To prevent this, use this option to disable AGC for these generators so that they will not move because of automatic MW control schemes.

Pricing Options

These options are available only in the OPF/SCOPF add on.

Solve Unconstrained Case

Select this option when you want an unconstrained solution to be obtained before an OPF or SCOPF solution for each time point.

Price Hydro Generation at Marginal Cost

During OPF and SCOPF simulations, hydro generation may experience large changes in output due to its low marginal cost. However, hydro generation is often not as cheap if limited water levels and dam restrictions are observed. These considerations are usually taken care of in the hydro-thermal coordination solution, outside of Simulator. In the OPF and SCOPF solutions it is important to assign a reasonable price to hydro generation to avoid large generation output deviations. A common mechanism to do that is to first obtain the system marginal cost, and then assign this cost to the hydro units.

Reset Hydro Gen Price at the End of Time Period

Choose this option to make the hydro generation price be reset for the next time point solution.

Save Binding Constraints

The binding constraints determined in the OPF/SCOPF are stored.

Time Step Simulation Options

The Time Step Simulation can be performed in two ways:

Continuous

With this option time points are solved one after another as soon as one time point finishes. The purpose of the simulation is to obtain the solutions for all time points as quickly as possible.

Timed

With this option the solutions are simulated as they would occur in actual time. The difference in date/time between two time points in the list defines a delay to start the solution of the next point. A **Time Scale** is used to set the speed of the simulation with respect to actual time. Suppose that you have 3 time points defined at the following date/times:

1/20/05 1:00 AM

1/20/05 2:00 AM

1/20/05 4:00 AM

Assume also that the **Time Scale** is 1 hour runs in 10 seconds. If you start the **Timed Simulation** you would see the conditions of the first time point applied to the power system immediately, the conditions of the second time point applied 10 seconds later, and those of the third point applied 20 seconds after the second point. The delays on the simulation allow you to see how the quantities evolve in actual time. In addition, you can animate the time simulation while each time point is being solved. The visualization of the **Timed Simulation** is enhanced when you use the [Time Step Simulation Toolbar](#).

Step Type

This section of options allows you to specify if the data for each time point should be applied only, or if the data should be applied and the load flow solved. Use **Apply and Solve** to apply the input data and actually solve the power flow. Use **Just Apply Data** to apply the input data and not solve the power flow. Check the **Apply Input Data** box to apply input data and custom input data. This does not include scheduled data. Check the **Apply Schedule Data** box to apply scheduled data.

Any pre-script or post-script commands will be applied regardless of how these options are set.

Auto Load TSB File Options (saved in case pwb file)

These are options that relate the power system case (.pwb file) to the time series binary file (.tsb file). These options are saved with the .pwb case.

Automatically Load Default *.tsb File

The .tsb file specified in the **Default *.tsb file** is loaded automatically when opening the .pwb case. If the file cannot be found, a message will be issued.

Automatically Run Simulation after Loading *.tsb

This option is only relevant if the .tsb file was automatically loaded. This instructs the Time Step Simulation form to start the simulation immediately following loading the .tsb file.

Automatically Set Default *.tsb File to Current *.tsb File

When leaving Simulator, the current time series information is saved in the **Default *.tsb file**.

Default *.tsb File

Default path and name of the *.tsb file.

Save the Default *.tsb File after Finishing the Run

If this option is checked, immediately upon finishing the simulation run, the settings and results are saved in the **Default *.tsb file**.

Result Storage Options

These options allow the selection of where the results from the simulation are directed.

Store results in memory

Only store the results in the computer's random access memory (RAM). This option is prone to running out of memory when many fields are stored over many time points.

Only store results in csv file

With this option the results are sent directly to a CSV file without filling up computer memory. Keep in mind that when this option is chosen, the [Results grids](#) will not contain any data.

Store in memory and csv file

With this option the results will be stored both in the computer's memory and sent to CSV file.

CSV Object ID Options

When choosing to store results in a CSV file, this option will determine if the objects will be identified by **Primary key (number)**, **Secondary key (name)**, or **Label**.

CSV File Identifier

This is a label that will be prepended to every generated CSV file if choosing to send results to CSV. Files are generated for each result type. For example if a user is storing bus voltage and angle, the results will be stored in a file named *CSV File Identifier_Buses.csv*. A separate file will be created for each of the results types to store data of that type.

CSV Output Path

This specifies the computer or mapped network drive where the CSV results will be stored.

Auto Contouring Options

The Time Step Simulation allows you to contour quantities on the oneline diagram using Simulator [Contouring](#) at each time point. Optionally, these contour diagrams can be saved in different formats.

No Auto Contouring

Contouring is not used during the Time Step Simulation. Although using the **Timed Simulation** you can see the quantities change and the animation take place on the diagrams, the contouring is not displayed.

Contour but Do not Save

Contouring takes place at each time point, but the diagrams are not saved.

Save in File as Bitmap

The contouring diagrams generated at each time point are saved in Bitmap format.

Save in File as JPEG

The contouring diagrams generated at each time point are saved in JPEG format.

Contour File Name Format

The Bitmap or JPEG contouring diagrams are saved using the specified format, which includes the time point date/time.

Run Pre-Script Command

This option allows the pre-script command to be run Before Applying Input Data or Right After Applying Input Data. Input data includes any hourly input data and scheduled actions. With either setting, the pre-script command is applied prior to solving the power flow solution for the time point.

Run Post-Script Command

This option allows the post-script command to be run either Before Storing Results or After Storing Results. Running the post-script command before storing the results could be useful in situations in which sensitivity analysis is done based on the solution of the time point and these sensitivities need to be stored as part of the results for that time point. Running the post-script command after storing the results could be useful in situations in which a specific control setting was used for the current time point and needs to be restored to the original setting before moving on to the next time point.

Set Reference Case

The reference case is the case that will be restored when resetting the simulation or starting a simulation run. The reference case is initially set to be the case in memory when Time Step Simulation dialog is first opened. If changes are made to the case in memory while the Time Step Simulation dialog is open and these changes need to be reflected in the reference case, click this button to set the case that is currently in memory to be the reference case.

Time Step Simulation: Specifying and Maintaining a List of Timepoints

The list of time points is the basis for the [Time Step Simulation](#). Simulator will go through the list of time points and solve each one of them in sequence. Results will be available only for those time points specified in the list. Intervals as small as one second can define a time point.

Whether you start with an empty list or you already have time points in it, you can insert new time points by right-clicking in the **Summary** page and selecting **Insert New Timepoint(s)**. This will bring up the [New Time Point Dialog](#).

The list of time points will always be sorted based on the date/time shown on the Date and Time columns in the [Summary page](#). Thus, if you create a new time point with an intermediate date/time, Simulator will insert it at the appropriate place in the list. If you need to change the date/time of a time point you can right-click in the **Summary** grid and select **Time Point records > Change Timepoint Time** to bring up the [Change Timepoint Time Dialog](#). This dialog is similar to the [New Timepoint Dialog](#), with the exception that instead of specifying the date/time for a new time point, you will be modifying the date/time of an existing time point. If the date/time matches the date/time of an existing time point, a warning message is issued.

Each time point is linked to its results data in what is called a **Timepoint Record**. The **Timepoint Record** contains the date and time of the time point, all the input data specified for that point, and if any, the results that have been obtained for that time point. If you delete a time point, the entire record is deleted with it. In order to delete the time point you can right-click in the **Summary** grid and select **Delete Entire Timepoint Record**. Note that you can vertically select cells in this grid and delete several time point records at a time.

Suppose that you have 1:00 AM and 3:00 AM time points that have associated with them hourly MW load data. If you insert 2:00 AM data, the 2:00 AM cell of the MW load will appear empty. If you run a study, no data will be applied to the power system at 2:00 AM, but you will get a result (identical to the one of 1:00 AM). You need to fill the 2:00 AM cells with data in order for the values to be applied at that time point and obtain the correct results for 2:00 AM.

Time Step Simulation: New Timepoint Dialog

The New Timepoint Dialog is used to define new time points for a [Time Step Simulation](#). This dialog is accessed from the [Time Step Simulation dialog](#) by right-clicking on any of the grids that displays time points and selecting **Insert New Timepoint(s)** from the local menu or clicking the **Insert Time Points** button at the top of the dialog. One or multiple time points can be defined at once. Use the date drop-down box to bring up a calendar for easy selection of the date. Note that the calendar has visual controls that allow you to navigate through months or years, which allows you to set the desired date easily.

If this is the first time point you will insert in the list, by default the dialog is populated with today's date at 12:00 AM. If there are other points in the list, the default is one hour after the date/time of the last time point.

The other options in this dialog are:

Date

Use this control to select the date of the time point. You can use the visual controls or the up or down arrow keys to modify the date.

Time

Use the control to enter the hour, minute, second, and AM/PM description of the time. Once you are positioned on the hour, minute, second, or AM/PM values, you can use the up/down arrow buttons for easy selection without having to type. Note that the time points are specified with a precision of up to 1 second.

Total Number of Timepoints to Enter

By default this field is set to 24, meaning that you will enter an entire day's worth of time points with the specified date and time if the interval is left at the default specification of one hour.

New Timepoint Interval: Hours

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the hour(s) of the interval between each time point. Time points will be Hours + Minutes + Seconds apart. Default is one hour.

New Timepoint Interval: Minutes

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the minute(s) of the interval between each time point. Time points will be Hours + Minutes + Seconds apart. Default is zero minutes.

New Timepoint Interval: Seconds

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the second(s) of the interval between each time point. Time points will be Hours + Minutes + Seconds apart. Default is zero minutes.

OK, Cancel

Click **OK** to close the dialog and insert the specified time points. Click **Cancel** to close the dialog and abandon any changes.

Time Step Simulation: Change Timepoint Time Dialog

The Change Timepoint Time Dialog is used to modify the date/time of a time point for the [Time Step Simulation](#). Only one time point time can be changed at a time. If the new time assigned to the time point already exists, a warning message is generated prompting for the input of a different date/time. If the new date/time does not exist, the time point will be moved to the correct position in the time point list.

The Change Timepoint Time Dialog is accessed by right-clicking on any of the Summary, Input, and Results [Time Step Simulation pages](#) that list the time point date/time and choosing **Time Point records > Change Timepoint Time** from the local menu.

Date

Click the calendar icon to use the calendar control to select the date of the time point. You can also position the mouse cursor over the month, day, or year and use the up/down keys on the keyboard to adjust these fields.

Time

Use the control to enter the hour, minute, seconds and AM/PM description of the time. Once the mouse cursor is positioned on the hour, minute, seconds or AM/PM value, you can use the up/down keys on the keyboard for easy selection without having to type. Note that the time points are specified with a precision of up to 1 second.

Time Step Simulation: TSB Case Description

This page is used to describe the time series binary file (.tsb file) for informational purposes. Use the memo box that comprises most of this page to comment on the case, .tsb file, time step simulation, or anything else that is relevant. These comments will be stored with the .tsb file. The **Version Used to Store the TSB File** and the **Simulator Build Date** are also provided.

Time Step Simulation: Insert/Scale Column Dialog

The Insert/Scale Column Dialog is opened from the [Input page](#) grids of the [Time Step Simulation dialog](#). These [matrix grids](#) are used to specify input data for MW Loads, Mvar Loads, Generator MW, Generator Maximum MW, Area Loads, Zone Loads, and Injection Groups. Time point data for each particular object is specified in columns. When no column has been added to the grid, the Insert/Scale Column Dialog allows you only to select the new object. When there are existing columns on the grid, the dialog allows you to insert a **New Column** either alone or based on the values of the existing columns. This option is available because values such as load MW data tend to experience similar fluctuations in time.

Current Column

If there are existing columns and the Insert/Scale Column Dialog is called from one of these existing input data columns, this field shows the position of that column. This field tells the user what column the **New Column** will be based on.

Action

These options determine how the new column will be inserted.

Scale Entire Current Column

This option is available only when the current column corresponds to an existing input data column. When applied, the new column takes the values of the current column scaled by the **Scaling Factor**.

Scale Selected Rows of Current Column

This option allows you to select a group of contiguous rows of a column and scale only those rows of the **Current Column** by the **Scaling Factor**.

Insert New Column Derived from Current Column

This option applies the **Scaling Factor** to the values of the of the **Current Column** and uses these scaled values to populate the **New Column**.

Scaling Factor

Factor used for scaling the current column, selected rows of the current column, or inserting a new column derived from the current column.

Load Scaling

These options are available only when inserting Load columns and are used to scale **Real and Reactive Load**, **Just Real Load**, or **Just Reactive Load**.

New Columns Selector

Use this selector to specify the ID of the object(s) that will be added as new column(s) to the grid. The bottom section of the dialog is a selector that allows you to specify one or multiple objects at a time and add columns for them by passing the objects from the left list to the right list. All of the options available with the selector may not be shown by default. Click on the down arrow located to the left of the Sort by option to open the menu that allows selection of which options to display. This selector has the following options:

Sort by Name

Check this option to sort the list of available objects by name

Sort by Number

Check this option to sort the list of available objects by number

Define/Find Filter

Press this button to filter the list of available objects using an [Advanced Filter](#).

Use Area/Zone Filters

Check this box to filter the list of available objects using the Area/Zone filters.

Search Next/Search All

By typing the start of the name of an object in the edit line, you can search the next object or all the objects available that match the search pattern. You can also use wildcards to search for objects.

List of Objects

The objects in the list are selected by clicking on them. Multiple objects that are together can be selected by clicking the mouse while holding the SHIFT key. Multiple objects that are not contiguous in the list can be selected by clicking the mouse while holding the CTRL key.

Select All/Clear All

Use these buttons to select or clear all items selected in the list of objects.

Select Button

Press the blue arrow button to pass the selected objects from the left list to the right list.

Remove Button

Press the trash can button to remove the objects selected in the right list.

For more information about methods to specify input data, please read the [Loading Input Data](#) section.

Time Step Simulation: Loading Input Data

The [Time Step Simulation](#) allows you to specify the operating conditions of your power system through input data. The input data can be of two types:

Scheduled Data, which is specified for data that more naturally spans multiple time points rather than being defined at each time point. To set up scheduled data see the [Setting up Scheduled Input Data](#) section.

Time Point-Based Data is specified for each time point in the [Input pages](#). There are several ways to specify input data:

- By entering data manually on the [Input pages](#).
- By deriving or scaling values from another column using the [Insert/Scale Column Dialog](#).
- By loading previously formatted data from Excel or .csv files through the **Read Buttons** in the [Time Step Simulation Dialog](#).
- By pasting data from Excel directly to the grid. A common way to do that is to:
 - o Set up the desired time points and data columns on the grids of the [Input pages](#)
 - o Copy the template to Excel by right-clicking and selecting **Copy/Paste/Send > Send All to Excel** on the local menu
 - o Fill the Excel sheet with the appropriate data
 - o Paste the data back to the data grid by selecting and copying it in the Excel sheet, including the column headers, and then right-clicking on the grid back in Simulator and selecting **Copy/Paste/Send > Paste** on the local menu

Time Step Simulation: Setting up Scheduled Input Data

There are two types of input data for the [Time Step Simulation](#):

Time Point-Based Data, which is described in the [Loading Input Data](#) section; and,

Scheduled Data, which is specified for data that more naturally spans multiple time points rather than being defined at each time point. In this section we describe how to specify this type of input data.

Although it is possible to specify the operating conditions of a power system exclusively by quantities at each time point, there are several quantities whose specification would be redundant and would require significant memory storage if defined in this manner. Examples of such quantities are:

- The status of a transmission line that is taken out of service on a particular date and time for maintenance.
- The status of a generator, which follows a particular maintenance schedule.
- A generator's voltage set point that is different during the day or at night.
- A scheduled transaction between areas that has different MW set points applied at 10 am, 4 pm and 10 pm.
- A capacitor connection status for a Mvar block that is used only during the day.
- An industrial load that operates at different levels for different shifts.
- A peaker generating unit that operates only during certain hours of the day
- Any many others

All of these quantities can be specified by introducing the concept of **Schedule**. A schedule is a list of pairs (**Date Time, Value**), where the value can be numerical, conditional (Yes/No or Closed/Open) or text. The Schedule can have any number of time points and can be periodic. The schedule defines the "shape" of how a quantity varies in time. In order to define a schedule, go to the [Schedules sub-page](#) of the [Input page](#), right-click on the grid and select **Insert New Schedule** to bring up the [Schedule Dialog](#).

Once a schedule has been created, we can assign an object field, such as the status of a transmission line or the MW output of a generator to the schedule by means of a **Schedule Subscription**. The object field will follow the schedule "shape" in time. The use of Schedule and Schedule Subscription objects gives us great flexibility in specifying how quantities should vary. In particular, it is possible to assign many fields to the same schedule. In order to define a **Schedule Subscription**, go to the [Sched Subscriptions sub-page](#) of the [Input page](#), and right-click on the grid and select **Insert New Subscription** to bring up the [Schedule Subscription Dialog](#).

An important feature of the Schedules is that their date/times do not need to match the date/times of the list of time points (time points listed in the [Summary page](#)). Suppose that your list of time points are defined hourly for the next day: 1am, 2am, etc. up to 11pm. You can schedule a particular action to take place at 2:35 am and see the results of that action in the next time point, i.e., 3 am. A special logic takes care of applying scheduled actions at appropriate time points asynchronously.

Time Step Simulation: Schedule Dialog

The Schedule Dialog is opened from the [Schedules page](#) of the [Time Step Simulation dialog](#) by right-clicking on the Schedules grid and choosing **Insert New Schedule** or **Show Dialog** for an existing schedule. This dialog is used to define or modify a schedule. A schedule is a list of time points together with a numeric, conditional (Yes/No or Closed/Open) or text value that defines the "shape" of how a quantity should vary in time. The time points are listed on the grid section of the dialog. The dialog also contains a number of options that define the schedule.

When a schedule is created, the time step simulation inserts by default four time points for the current date with a numeric value of zero in every point. You can then change the date and times of the schedule time points and specify different values.

Schedule Dialog Local Menu Actions

The Schedule Dialog grid, as with any other Simulator grid has a local menu that is accessed by right-clicking on any cell. The most important options of this local menu are:

Insert New Point(s)

Select this option to bring up the **New Schedule Point Dialog**. This is the same as the [New Time Point Dialog](#) but is used here to insert Schedule time points.

SchedPoint records > Change Time

Select this option if you want to change the date/time of a schedule time point. The selection brings up the **Change Time Dialog**, which is the same [Change Timepoint Time Dialog](#) used here to change the date/time of a schedule time point.

Delete

Use this option to delete the current schedule time point.

Set/Toggle/Columns > Select Column(s)

Use this option to select an entire column. Note that you can use this option and then delete all the time points.

Copy/Paste/Send > Send All to Excel

Use this option to copy the values in the Schedule grid to Excel. You can then easily modify the values using Excel and then Paste back the new schedule points into Simulator.

Timepoint List Options

These options provide shortcuts to some of the local menu actions related to the maintenance of the list of Schedule time points:

Add Point

Adds a new time point at the end of the list. By default the date/time of the new time point is one hour after the last time point. If there are no time points in the list, a time point is inserted with today's date at 12:00 AM.

Delete Point

Deletes the selected time point(s).

Shift Date Time Buttons

Use these buttons to move by one week, day, hour, minute or second the date time of the selected time point.

Schedule Options

The upper part of the Schedule Dialog is used to define other schedule options:

Schedule Name

Enter the name you want to give to the schedule. This is how you will identify the schedule when you set up [schedule subscriptions](#). If the user does not specify otherwise, Simulator assigns names Sched1, Sched2, etc. as new schedules are created.

Name Suffix

This is a string automatically generated by Simulator, which is used to display the main characteristics of the Schedule together with the name. The Name Suffix is composed of three strings separated by periods:

[**Num**, **Y/N**, or **Txt**]: Indicates whether the schedule values are numeric, conditional or text.

[**NPER** or **PER***adbc***mds**]: The string indicates whether the schedule is non-periodic (NPER) or periodic (PER). If it is periodic, it indicates the days (a), hours (b), minutes (c), and seconds (d) of the period.

[**Nn**]: Indicates the number of time points (n) of the Schedule.

By looking at the Schedule suffix, the user effectively avoids writing long names for the Schedule to characterize its properties.

Value Type

Numeric, conditional (*Yes/No* or *Closed/Open*) or *Text*. When you toggle this selection, the value column in the Schedule grid changes its heading from **Numerical Value** to **Yes/No Value** to **Text Value**, and the numerical cells (blue) are changed to Yes/No cells (green), indicating that you can toggle the cell value, to blank for Text Value. Note that numeric fields, such as Load MW, Schedule Transactions MW etc., will subscribe to a Numeric Schedule, whereas fields such as Line Status will subscribe to a Yes/No Schedule. Text schedules can be subscribed to custom string and memo fields. For more details on how to subscribe to schedules, see the [Schedule Subscriptions](#) topic.

Interpolate Values Between Time Points

The purpose of creating a schedule is to modify data that does not need to be defined for each individual Time Step Simulation time point. When this option is checked, the value for a schedule will be interpolated for the current time point based on the values specified for the schedule time points.

Apply Schedule Points as Events

When this option is checked, schedule values will only be applied if the time point matches the schedule time point. The values will not change for time points that do not match a schedule time point. For example, assume that a schedule has time points at 6:00 AM with a value of 50 and 11:00 AM with a value of 75. Assume that a generator is subscribed to this schedule. Also, assume that the time step simulation contains time points for every hour between 1:00 AM and 11:00 PM. For the time points between 1:00 AM and 5:00 AM the output of the generator will remain at what is was in the base case, unless of course there is some other input data that changes the output of this generator. If there is some other input or manual intervention that changes the output of the generator, the output of the generator will not be changed again by the schedule until 6:00 AM. At 6:00 AM the output of the generator will change to 50. At 7:00 AM the output of the generator will not be updated and remain at 50, unless the output of the generator is changed by some other input or manual intervention. The same is true for time points 8:00 AM through 10:00 AM. At 11:00 AM the output of the generator will change to 75 because of the schedule.

When this option is not checked, the schedule will be applied at every time point. Assume the same example as above. For the time points between 1:00 AM and 5:00 AM the output of the generator will not be impacted by the schedule. At 6:00 AM the schedule will be applied and the output of the generator will change to 50. At 7:00 AM the schedule will be applied again and the output will again be set to 50. The same thing will occur at the time points between 8:00 AM and 10:00 AM. At 11:00 AM the schedule will be applied and the output will change to 75. At each time point after 11:00 AM the schedule will be applied at the output will change to 75. This means that if some other input or manual intervention causes the output of the generator to change between time points, at each time point the schedule will be applied and the output of the generator will be updated to match the schedule.

Periodic Options

A schedule becomes periodic when its values are repeated every certain period specified in days, hours, minutes and/or seconds. A logic condition for a schedule to be periodic is that its period be larger than the time span between the date/time of the last time point and the first time point in the schedule. For instance, if the schedule has three time points at 1:00 AM, 2:00 AM and 4:00 AM, the span is 3 hours. Thus the schedule period must be 4 hours or more. If this condition does not hold, Simulator issues a warning message.

Repeat Every

Check this box to make the schedule periodic. The schedule must have a total period greater than 0 to be considered periodic. The schedule period is the number of days plus the number of hour plus the number of hours plus the number of seconds specified.

Days

A positive integer that specifies the number of days of the schedule period.

Hours

A positive integer from 0 to 23 that specifies the number of hours of the schedule period.

Minutes

A positive integer from 0 to 59 that specifies the number of minutes of the scheduled period.

Seconds

A positive integer from 0 to 59 that specifies the number of seconds of the scheduled period.

Valid From

When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system before this date. By default, the Valid From date is set to Jan 01, 2000, at 12:00 AM.

Valid Until

When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system after this date. By default, the Valid Until date is set to Dec 31, 2030, at 12:00 AM.

Time Step Simulation: Schedules

The Schedules sub-page of the [Time Step Simulation dialog Input page](#) is used to display all of the defined schedules and their properties. Schedules are used to define input which is specified for data that more naturally spans multiple time points rather than being defined at each time point. Schedules define the "shape" of how a quantity should vary in time.

The most relevant options of the local menu, obtained by right-clicking on the grid of this page are:

Schedules Page Local Menu Actions

Insert New Schedule

Selecting this option will open the [Schedule Dialog](#) which allows the definition of a new schedule.

Delete

Select this option to delete the current Schedule.

Show Dialog

Select this option to bring up the [Schedule Dialog](#) with the information of the current Schedule.

Schedule Page Fields

The Schedules sub-page contains the following information about schedules:

Name

Name of the schedule.

Suffix

Schedule suffix used to provide a short-hand description of how the schedule is defined. For a description on how the suffix is created, please see the [Schedule Dialog](#) section.

Periodic

Indicates whether or not the schedule periodically repeats for a specified period of time (defined in days, hours, minutes, and/or seconds).

Period Days, Hours, Minutes, and Seconds

If the schedule is periodic, number of days, hours, minutes, and seconds in the schedule period.

Valid From

Set this to YES if the schedule is valid only from a specified starting date/time.

Period Start Date

If specifying a start date/time from which the schedule is valid, this is the start date.

Period Start Hour

If specifying a start date/time from which the schedule is valid, this is the start time.

Valid Until

Set this to YES if the schedule is valid only up to a specified ending date/time.

Period End Date

If specifying an end date/time up to which the schedule is valid, this is the end date.

Period End Hour

If specifying an end date/time up to which the schedule is valid, this is the end time.

Time Step Simulation: Schedule Subscription Dialog

This dialog is used to define and display the options of a Schedule Subscription. Schedule Subscriptions tell the Time Step Simulation that a specific object field (Gen MW, Line Status, Scheduled Transaction MW, etc.) should vary according to the "shape" specified in a Schedule. This dialog can be accessed from the local menu of the [Schedule Subscriptions page](#) of the [Time Step Simulation dialog](#).

Simulator objects have a large number of fields, which can be classified as numeric, boolean, or text. Numeric fields, whether integer or real, can subscribe to Numeric Schedules, boolean fields can subscribe to Yes/No Schedules, and text fields can subscribe to Text Schedules. Thus, a condition for a field to subscribe to a schedule is that a schedule of its type be already defined.

The dialog includes the following options:

Active

Indicates that the schedule subscription is active. If this box is not checked, the schedule subscription is defined, but no schedule action will be applied to the power system.

Object Type

Use this selector to specify the object type that you want to subscribe to a schedule. Currently, Simulator supports Generators, Loads, Line/Transformers, Shunts, Areas, Transactions, Zones, Interfaces, and DC Lines. When you select an **Object Type**, three things happen:

- The **Object ID** list is populated with the elements of the object type present in the power flow case and the first element is displayed by default.
- The **Field** selector is populated with the enterable fields corresponding to that particular **Object Type** and the first field is displayed by default.
- The type of the first field (Numeric, Yes/No or Text) is identified and the **Schedule** selector is populated with the existing schedules of that type.

Object ID

Use this selector to specify the particular object of a particular **Object Type** whose field you want to subscribe to a schedule. The **Object ID** has above it a string that indicates how the ID of the object is built depending on the **Object Type**. For instance, when the **Object Type** is Generator, the **Object ID** is specified as the Bus Number and then the Gen ID.

Select/View Objects

Press this button to bring up the Add Schedule Subscribers Dialog. This dialog is used to select multiple objects of the specified type, for which a field will subscribe to the schedule. When multiple objects are selected, the Object ID indicates "Multiple Objects" followed by the number of objects that will be selected. If there are objects whose fields are subscribed to a schedule, these objects will appear in the right side selection list, because they are currently subscribing to the schedule.

Field

Use this selector to indicate the particular **Field** that you want to subscribe to a schedule. Depending on the type, the Dialog indicates whether the **Field** should subscribe to a Numeric, Yes/No, or Text schedule. Once you have selected the **Field**, the **Schedule** selector is populated only with the schedules of that particular type.

Schedule

Use this selector to specify the schedule to which the **Field** should be subscribed.

Time Shift

The time shift options can be used to Delay or Advance applying the scheduled actions by the specified **Delay Days**, **Delay Hours**, **Delay Minutes**, or **Delay Seconds**. For instance, suppose that you have setup a schedule so that a new 100 MW generating unit enters online on March 10th. The entering schedule is complex and consists of increasing the output in steps of 20 MW each hour starting at 10 am. Suppose that due to unexpected events, the connection of the unit must be put off by two days. Instead of modifying the entire schedule, you could set a 2 day *Delay* so that the unit enters online on March 12th at 10 am.

If instead the new unit can go online ahead of schedule on March 8th, the Time Shift can be set to *Advance* for 2 days so that the unit will go online earlier.

Subscription Type

In addition to the time shift, the value "shape" defined by a numeric schedule can be altered by making the subscription **Relative**.

Absolute

Select this option so that the field takes the exact value specified in the schedule.

Relative

The numeric values of the schedule can be altered by a **Multiplier** and a **Value Shift**. In this linear modification, the value of the field that is applied to the power system will be equal to:

$$\text{Actual Field Value} = \text{Multiplier} * \text{Schedule Value} + \text{Value Shift}$$

Time Step Simulation: Schedule Subscriptions Page

This sub-page is found on the [Input page](#) of the [Time Step Simulation dialog](#) and is used to display all of the schedule subscriptions and their properties. Once a [schedule](#) has been created, we can assign an object field, such as the status of a transmission line or the MW output of a generator to the schedule by means of a Schedule Subscription. The object field will follow the schedule "shape" in time. The use of Schedule and Schedule Subscription objects gives us great flexibility in specifying how quantities should vary.

The most relevant options of the local menu, obtained by right-clicking on the grid of this page are:

Schedule Subscriptions Page Local Menu Actions

Insert New Subscription

Use this option to define a new schedule subscription through the [Schedule Subscription Dialog](#).

Delete

This option deletes the current schedule subscription.

Show Dialog

This option brings up the [Schedule Subscription Dialog](#) with the information of the current schedule subscription.

Schedule Subscriptions Page Fields

The Schedule Subscriptions Page shows the following information:

Object

Object Type: Generator, Load, Line/Transformer, Shunt, Area, Transaction, Zone, Interface, or DC Line

Object IDs

Are explained in the following table:

Object Type	Object ID1 <i>(Numeric)</i>	Object ID2 <i>Numeric</i>	Object ID3 <i>String[2]</i>
Generator	Bus Number		Gen ID
Load	Bus Number		Load ID
Line/Transformer	From Bus Number	To Bus Number	Circuit ID
Shunt	Bus Number		Shunt ID
Area	Area Number		
Transaction	From Area Number	To Area Number	ID
Zone	Zone Number		
Interface	Interface Number		
DC Line	Rectifier	Inverter	ID

Number

Number

Object Field

Field that subscribes to the schedule.

Schedule Name

Name portion of the schedule name and suffix identifier.

Schedule Suffix

Suffix portion of the schedule name and suffix identifier.

Active

Set this to YES so that the schedule values will actually be applied to the power system.

Relative

If this is NO the field takes the exact schedule values. If this is YES the schedule takes the value of the schedule scaled by a multiplier plus Value Shift.

Multiplier

Scaling factor used when the subscription is Relative.

Value Shift

Shift value used when the subscription is Relative.

Delay/Advance?

Indicates if the schedule should be delayed or advanced to a new point indicated by the shift values. A delay will cause the schedule to start later in time. An advance will cause the schedule to start earlier in time. See the [Schedule Subscription Dialog](#) for more details on this option.

Day Shift

Number of days of the schedule time delay/advance.

Hour Shift

Number of hours of the schedule time delay/advance.

Minute Shift

Number of minutes of the schedule time delay/advance.

Second Shift

Number of seconds of the schedule time delay/advance.

Schedule Subscriptions can be saved and loaded from auxiliary files using options found on the local menu obtained by right-clicking on the table or using the case information toolbar.

Time Step Simulation: Application of Controller Time Delays

During a Time Step Simulation run, special control actions can be implemented for [transformers](#) and [switched shunts](#). These control actions allow time delays to be used so that a specified period of time passes, while a device is outside of its regulation range, before that device can implement a control change that will attempt to bring the device back within its regulation range. Multiple transformers and switched shunts can be on this type of control.

In addition to transformers and switched shunts that allow switching time delays during a time step simulation run, there are special [Time Step Actions](#) that can be utilized during a time step simulation run. Time Step Actions perform some action (e.g. opening a branch, moving load, changing generator output, etc.) once a specified Model Criteria evaluates to true and remains true for the specified time delay.

Time delays will only be used in a time step simulation if doing a complete time step simulation run. A complete run is one that is started using the Do Run button on the [Time Step Simulation dialog](#) and is allowed to run to completion without running any time steps out of order or repeating time steps. If not doing a complete time step simulation run, any transformers and switched shunts that model time delays will ignore the time delay settings and operate based on all other relevant control settings and Time Step Actions will be ignored.

When doing a complete time step simulation run, application of time delays fits into the time step simulation process in the following manner:

- Start the time step simulation by clicking the Do Run button on the Time Step Simulation dialog
- Initialize time delay objects and check for data errors
 - If any data errors result, provide user option to continue with Simulator fixing the errors or abort the run so that the user can fix the errors
 - Transformers and switched shunts that are modeling switching delays are turned off automatic control so that they will not move during the power flow solutions
- Process a time point
 - If applying pre-script command before applying input data, run pre-script command
 - If **Apply Input Data**
 - Update loads, MW and Mvar
 - Update generators, MW and Max MW
 - Update line statuses
 - Update area loads, MW and Mvar
 - Update zone loads, MW and Mvar
 - Update injection group injections
 - Apply custom inputs
 - If **Apply Schedule Data**
 - Apply schedule actions
 - If Input Data and Schedule Data can be applied with no errors or not applying these at all
 - If applying pre-script command after applying input data, run pre-script command
 - If choosing to **Apply and Solve**
 - Solve power flow
 - If successful power flow solution
 - If any objects modeling time delay, check for time delay implementation
 - [Transformers](#)
 - [Switched Shunts](#)
 - [Time Step Actions](#)
 - If any system changes, solve power flow

- If still successful power flow solution (time delay actions implemented and power flow solves or no time delay actions implemented)
 - If **Solution Type** <> *Single Solution*, run unconstrained OPF, OPF, or SCOPF
 - If **Run Contingencies** = *YES*, run contingency analysis
- If applying post-script command before storing results, run post-script command
- Store results
- If applying post-script command after storing results, run post-script command
- Continue processing time points until all completed
- Restore time delay objects to original control modes

Switched Shunt Control: Time Step Options

Switched Shunt Time Step Options are found on the **Time Step Options** subtab of the Control Options tab of the Switched Shunt Information dialog in either [Edit](#) mode or [Run](#) mode. These options can be modified if the switched shunt is on either discrete or continuous automatic control. The options are only used as part of the [Time Step Simulation](#) tool.

These options allow the specification of time delays between the time when a switched shunt is considered in violation of its regulation range and the time when the switched shunt nominal Mvar is changed by stepping through Mvar blocks in an attempt to bring the switched shunt back within its regulation range. Because time delays are involved, it is only possible in Simulator to use this type of control during a Time Step Simulation when there is an accounting of system changes over time.

The options described below dictate how an individual switched shunt using time delays will be evaluated during a time step simulation. For a description of the overall process of how switching time delays and time delay actions are applied during a time step simulation run, see the [Time Step Simulation: Application of Controller Time Delays](#) topic.

The following options are available for specifying switched shunt time delays:

Model Switching Delay in Time Step Simulation

Check this box to use the time delay during a time step simulation AND indicate that this switched shunt should be on control during the time step simulation. Simulator only allows one switched shunt at a given bus to be on automatic control. During the time step simulation, multiple shunts are allowed to be on control, but some special handling must be done for this to happen. For a switched shunt to be on control during the time step simulation, at least one switched shunt at its bus must have its Control Mode set to *Discrete* or *Continuous* and that switched shunt must also have its **Model Switching Delay in Time Step Simulation** option checked. All of the other switched shunts must have their Control Mode set to *Fixed*. Fixed switched shunts that also have their **Model Switching Delay in Time Step Simulation** option checked will be on control during the time step simulation. The control mode for the switched shunt that is not fixed will dictate the control mode for all of the fixed switched shunts during the time step simulation. If a switched shunt is allowed to be on control during the time step simulation and a time step simulation is processing, the switched shunt nominal Mvar setting will only be allowed to change (step through Mvar blocks) if the switched shunt is violating its regulation range for the specified period of time. During a power flow solution outside of the time step simulation, the switched shunt will operate in the usual manner based on all relevant control settings without implementing any time delay in the switching.

When this box is not checked, the switched shunt will operate in the usual manner based on all relevant control settings without implementing any time delay in the switching during the time step simulation and any power flow solutions outside of the time step simulation.

During the time step simulation, switched shunt switching delays will only be implemented if doing a complete time step simulation run. A complete run is one that is started using the Do Run button on the [Time Step Simulation dialog](#) and is allowed to run to completion without running any time steps out of order or repeating time steps.

The Regulation Low Value and High Value specified on the Parameters tab of the Switched Shunt Information dialog are used to define the primary regulation range when modeling a switching delay.

At each time step, the regulated value of the switched shunt will be checked to determine if it falls outside of the regulation range. If so, a timer will be started to determine how long the value is outside of the regulation range. If the timer has already been started and the regulated value is still outside and on the same side of the regulation range, the appropriate time delay, as described

below, will be used to determine if the switched shunt has been outside its regulation range for the specified amount of time and is now allowed to start stepping through Mvar blocks to bring the regulated value back in range. Once the switched shunt is allowed to start switching, the timer will be reset and the appropriate time delay will be used to determine when the next move can occur if the regulated value continues to remain outside the regulation range and on the same side of the range. Once the regulated value falls inside the regulation range or the regulated value switches sides of the regulation range, the timer will be reset.

Maximum Steps to Move

Once a switched shunt has been outside of its regulation range for the specified period of time and is actually allowed to switch, this is the maximum amount of steps that it is allowed to move at the current time step even if additional steps would be required to bring the switched shunt back within its regulation range.

First Move Delay (Below and Above) (secs.)

These two options specify the amount of time to wait in seconds before the switched shunt is allowed to switch if its regulated value falls outside of the regulation range. These values must be specified in whole seconds with the minimum delay being 1. These delays are used when the regulated value first falls outside its regulation range and the switched shunt has not started to switch. Once the switched shunt has started to switch, but the regulated value still remains outside its regulation range on the same side of the range and needs to continue stepping through Mvar blocks, the Next Move Delays will be used. Once the regulated value falls inside the regulation range or the regulated value switches sides of the range (i.e. it was below the range and is now above), the First Move Delays will be used if the regulated value again falls outside the regulation range.

The Below entry specifies the delay to use when the regulated value is below the regulation range. The Above entry specifies the delay to use when the regulated value is above the regulation range.

Next Move Delay (Below and Above) (secs.)

These two options specify the amount of time to wait in seconds before the switched shunt is allowed to switch if its regulated value falls outside of the regulation range. These values must be specified in whole seconds with the minimum delay being 1. These delays are used once the switched shunt has started to switch but the regulated value remains outside of the regulation range on the same side of the range and needs to continue stepping through Mvar blocks. The Next Move Delay will continue to be used for subsequent delays as long as the regulated value remains outside of and on the same side of the regulation range. Once the regulated value falls inside the regulation range or the regulated value switches sides of the range, the First Move Delays will be used if the regulated value again falls outside of the regulation range.

The Below entry specifies the delay to use when the regulated value is below the regulation range. The Above entry specifies the delay to use when the regulated value is above the regulation range.

Use Secondary Regulation Range

Check this box to specify a secondary regulation range that is different than the primary range specified with the switched shunt. The secondary range can only be used if the primary range is also used (i.e. Model Switching Delay in Time Step Simulation is checked). The expectation is that this range will be larger and the delays would be shorter to model the fact that the controls will act more quickly to a very large deviation.

To properly define the regulation ranges, one of the ranges must be completely contained within the other range. This is necessary to prevent conflicting control actions in which one range would require nominal Mvar changes to increase the regulated value while another range would require nominal Mvar changes to decrease the regulated value.

The secondary regulation range works somewhat independently of the primary regulation range. At each time step, the secondary regulation range is checked first. If the regulated value falls outside of the secondary regulation range, a timer is started for the secondary regulation range. If the timer for the secondary regulation range has already been started, the regulated value is still outside of the secondary regulation range and on the same side of the range, and the appropriate secondary time delay is met such that the switched shunt is allowed to switch, the timers for both the primary and secondary regulation ranges are reset and whether or not the primary regulation range is violated is not checked at that time step. If the secondary regulation range timer has not exceeded the appropriate time delay or the regulated value is within the secondary regulation range, the primary regulation range is checked to see if the regulated value is outside of that range. If the regulated value falls outside of the primary regulation range, a timer is started for the primary regulation range. If the timer for the primary regulation range has already been started, the regulated value is still outside of the primary regulation range and on the same side of the range, and the appropriate primary time delay is met such that the switched shunt is allowed to switch, the timers for both the primary and secondary regulation ranges will be reset. Once the switched shunt has started stepping through Mvar blocks for either of the regulation ranges and continues to remain outside one or both regulation ranges and on the same side of the appropriate range, the Next Move Delay, appropriate for the given range, will be used by both ranges to determine when the next move can occur.

Secondary Minimum and Secondary Maximum

Use these fields to specify the secondary regulation range.

Secondary Max Steps to Move

Once a switched shunt has been outside of its secondary regulation range for the specified period of time and is actually allowed to switch, this is the maximum amount of steps that it is allowed to move at the current time step even if additional steps would be required to bring the switched shunt back within its secondary regulation range.

Sec. First Move Delay (Below and Above) (secs.)

When using the secondary regulation range, define the time delays in the same manner as described above for the primary regulation range in **First Move Delay (Below and Above) (secs.)**.

Sec. Next Move Delay (Below and Above) (secs.)

When using the secondary regulation range, define the time delays in the same manner as described above for the primary regulation range in **Next Move Delay (Below and Above) (secs.)**.

Processing Switched Shunts

At each time step, all switched shunts that are allowed to be on control during the time step simulation are processed to determine if their regulated values are outside of their regulation range. If they are and the time delay has been met such that the switched shunt is allowed to switch in an attempt to bring the regulated value back inside the regulation range, that switched shunt is marked for further processing. The options specified above determine the outcome of this initial check.

Further processing marches through the list of switched shunts that are allowed to switch one-by-one. They are processed in the order of ascending bus number with only one switched shunt at a bus allowed to be on control at a time. If there are multiple switched shunts at a bus that are allowed to be on control during the time step simulation, these are processed one-by-one in iterative steps. For each iteration of this process, the one switched shunt at each bus that is allowed to be on control will be checked to determine if its regulated value is still outside of its regulation range and adjust its Mvar injection if necessary. If any adjustments are made, an estimate is made of any impact on system voltages and the voltages are updated before the next switched shunt in the list is

processed. In this way, an estimate is made of the impact that one switched shunt adjustment has on the others. This process is repeated with only one switched shunt at a given bus allowed to be on control at a time until all switched shunts that are allowed to be on control during the time step simulation have been processed.

As an example, assume that at a given time step there are 6 switched shunts with regulated values outside of their regulation ranges and the necessary time delays have passed so that an attempt needs to be made to adjust these switched shunts to bring the regulated values back in range. Shunt A is at bus 1, shunts B and C are at bus 2, and Shunts D, E, and F are at bus 3. All shunts that are not being processed because of their time delay being met are set to be on fixed control. Shunts A, B, and D are set to their specified control mode while shunts C, E, and F are set to fixed. The shunts on control are then processed in the manner described above that determines the shunt adjustments and estimates new voltages. Shunts C, E, and F still have not been processed. Shunts A, B, and D are set to fixed with shunts C and E being set to their specified control modes. These shunts are then processed in the order of ascending bus number as described above. At the next iteration, all of the shunts are fixed except shunt F. This process will continue with only a single shunt at a given bus on control until all shunts that have met their time delay have been processed.

Time Step Actions

Time Step Actions are special actions that can be implemented during a [Time Step Simulation](#) run. These actions are very similar to contingency actions except that a model criteria and time delay must be specified. During a time step simulation run, a time step action will be implemented if its model criteria evaluates to true for the specified time delay.

At each time step, the model criteria for a time step action is evaluated. If it evaluates to true, a timer is started to determine how long the model criteria evaluates to true. If at any time step the model criteria evaluates to false, the timer will be stopped. If the timer has already been started, the model criteria still evaluates to true, and the specified time delay has been met or exceeded, the action will be implemented. Once a time step action is implemented during a time step simulation run, the action cannot be implemented again for the duration of that time step simulation run.

During the time step simulation, time step actions will only be evaluated and implemented if doing a complete time step simulation run. A complete run is one that is started using the Do Run button on the [Time Step Simulation dialog](#) and is allowed to run to completion without running any time steps out of order or repeating time steps. Time step actions are not currently used with any other tools.

The above describes how an individual time step action will be evaluated during a time step simulation run. For a description of the overall process of how switching time delays and time delay actions are applied during a time step simulation run, see the [Time Step Simulation: Application of Time Delays](#) topic.

Time step actions can be created or modified by accessing the Time Step Actions display available on the [Model Explorer](#) in the Solution Details folder or the [Input page](#) of the Time Step Simulation dialog. This display is a type of [Case Information Display](#) and has the abilities common to this type of display. To create a new time step action, right-click on the display and choose **Insert**. To modify an existing record, either edit the information directly in the table or right-click and choose **Show Dialog** to bring up the Time Step Actions dialog.

The Time Step Actions dialog is very similar to the [Contingency Element dialog](#). Specifying **Element Type** and **Action Type** for time step actions is done in exactly the same manner as for contingency element actions, with a few exceptions. Time step actions do not allow the Element Types of *Solve Power Flow* or *Contingency Block* to be included. Time step actions must specify a **Model Criteria**. A time step action cannot be created without a valid model criteria. If at some point prior to the time step simulation run a model criteria gets deleted, a time step action using that model criteria will not be evaluated during the time step run. The **Time Delay** must also be specified for a time step action. The Time Delay is specified in whole seconds with the minimum allowed being 1. Valid **Status** options are *CHECK* and *NEVER*. Set the Status to *NEVER* to prevent a time step action from being evaluated and implemented during a time step simulation run.

Transformer Control: Time Step Options

Transformer Time Step Options are found on the **Time Step Options** tab of the Transformer Control Info dialog (this dialog is different based on the automatic control types of [Voltage Regulation \(AVR\)](#), [Reactive Power Control](#), and [Phase Shift Control](#)). These options can be modified if the transformer is on some sort of automatic control. The options are only used as part of the [Time Step Simulation](#) tool.

These options allow the specification of time delays between the time when a transformer is considered in violation of its regulation range and the time when the transformer tap settings are actually changed in an attempt to bring the transformer back within its regulation range. Because time delays are involved, it is only possible in Simulator to use this type of control during a Time Step Simulation when there is an accounting of system changes over time.

The options described below dictate how an individual transformer using time delays will be evaluated during a time step simulation. For a description of the overall process of how switching time delays and time delay actions are applied during a time step simulation run, see the [Time Step Simulation: Application of Time Delays](#) topic.

The following options are available for specifying transformer time delays:

Model Tap Delay in Time Step Simulation

Check this box to use the time delay during a time step simulation. If this option is checked, the transformer is on automatic control (transformer must allow automatic control at the device, area, and global level), and a time step simulation is processing, the transformer tap (phase angle if a phase shifter) setting will only be allowed to move if the transformer is violating its regulation range for the specified period of time. During a power flow solution outside of the time step simulation, the transformer will operate in the usual manner based on all relevant control settings without implementing any time delay in the switching.

When this box is not checked, the transformer will operate in the usual manner based on all relevant control settings without implementing any time delay in the switching during the time step simulation and any power flow solutions outside of the time step simulation.

During the time step simulation, transformer tap delays will only be implemented if doing a complete time step simulation run. A complete run is one that is started using the Do Run button on the [Time Step Simulation dialog](#) and is allowed to run to completion without running any time steps out of order or repeating time steps.

The Regulation Minimum and Maximum values specified on the Common Options tab of the Transformer Control Info dialog are used to define the primary regulation range when modeling a tap delay.

At each time step, the regulated value of the transformer will be checked to determine if it falls outside of the regulation range. If so, a timer will be started to determine how long the value is outside of the regulation range. If the timer has already been started and the regulated value is still outside and on the same side of the regulation range, the appropriate time delay, as described below, will be used to determine if the transformer has been outside its regulation range for the specified amount of time and is now allowed to start moving tap positions to bring the regulated value back in range. Once the transformer is allowed to move tap positions, the timer will be reset and the appropriate time delay will be used to determine when the next move can occur if the regulated value continues to remain outside the regulation range and on the same side of the range. Once the regulated value falls inside the regulation range or the regulated value switches sides of the regulation range, the timer will be reset.

Maximum Steps to Move

Once a transformer has been outside of its regulation range for the specified period of time and is actually allowed to change tap positions (phase angle if a phase shifter), this is the maximum amount of steps that it is allowed to move at the current time step even if additional steps would be required to bring the transformer back within its regulation range.

First Move Delay (Below and Above) (secs.)

These two options specify the amount of time to wait in seconds before the transformer is allowed to change tap positions (phase angle if a phase shifter) if its regulated value falls outside of the regulation range. These values must be specified in whole seconds with the minimum delay being 1. These delays are used when the regulated value first falls outside its regulation range and the transformer has not started to change tap positions. Once the transformer has started to change tap positions, but the regulated value still remains outside its regulation range on the same side of the range and needs to continue changing tap positions, the Next Move Delays will be used. Once the regulated value falls inside the regulation range or the regulated value switches sides of the range (i.e. it was below the range and is now above), the First Move Delays will be used if the regulated value again falls outside the regulation range.

The Below entry specifies the delay to use when the regulated value is below the regulation range. The Above entry specifies the delay to use when the regulated value is above the regulation range.

Next Move Delay (Below and Above) (secs.)

These two options specify the amount of time to wait in seconds before the transformer is allowed to change tap positions (phase angle if a phase shifter) if its regulated value falls outside of the regulation range. These values must be specified in whole seconds with the minimum delay being 1. These delays are used once the transformer has started to change tap positions but the regulated value remains outside of the regulation range on the same side of the range and needs to continue changing tap positions. The Next Move Delay will continue to be used for subsequent delays as long as the regulated value remains outside of and on the same side of the regulation range. Once the regulated value falls inside the regulation range or the regulated value switches sides of the range, the First Move Delays will be used if the regulated value again falls outside of the regulation range.

The Below entry specifies the delay to use when the regulated value is below the regulation range. The Above entry specifies the delay to use when the regulated value is above the regulation range.

Use Secondary Regulation Range

Check this box to specify a secondary regulation range that is different than the primary range specified with the transformer. The secondary range can only be used if the primary range is also used (i.e. Model Tap Delay in Time Step Simulation is checked). The expectation is that this range will be larger and the delays would be shorter to model the fact that the controls will act more quickly to a very large deviation.

To properly define the regulation ranges, one of the ranges must be completely contained within the other range. This is necessary to prevent conflicting control actions in which one range would require tap changes to increase the regulated value while another range would require tap changes to decrease the regulated value.

The secondary regulation range works somewhat independently of the primary regulation range. At each time step, the secondary regulation range is checked first. If the regulated value falls outside of the secondary regulation range, a timer is started for the secondary regulation range. If the timer for the secondary regulation range has already been started, the regulated value is still outside of the secondary regulation range and on the same side of the range, and the appropriate secondary time delay is met such that the transformer is allowed to change tap positions, the timers for both the primary and secondary regulation ranges are reset and whether or not the primary regulation range is violated is not checked at that time step. If the secondary regulation range timer has not

exceeded the appropriate time delay or the regulated value is within the secondary regulation range, the primary regulation range is checked to see if the regulated value is outside of that range. If the regulated value falls outside of the primary regulation range, a timer is started for the primary regulation range. If the timer for the primary regulation range has already been started, the regulated value is still outside of the primary regulation range and on the same side of the range, and the appropriate primary time delay is met such that the transformer is allowed to change tap positions, the timers for both the primary and secondary regulation ranges will be reset. Once the transformer has started to move tap positions for either of the regulation ranges and continues to remain outside one or both regulation ranges and on the same side of the appropriate range, the Next Move Delay, appropriate for the given range, will be used by both ranges to determine when the next move can occur.

Secondary Minimum and Secondary Maximum

Use these fields to specify the secondary regulation range.

Secondary Max Steps to Move

Once a transformer has been outside of its secondary regulation range for the specified period of time and is actually allowed to change tap positions (phase angle if a phase shifter), this is the maximum amount of steps that it is allowed to move at the current time step even if additional steps would be required to bring the transformer back within its secondary regulation range.

Sec. First Move Delay (Below and Above) (secs.)

When using the secondary regulation range, define the time delays in the same manner as described above for the primary regulation range in **First Move Delay (Below and Above) (secs.)**.

Sec. Next Move Delay (Below and Above) (secs.)

When using the secondary regulation range, define the time delays in the same manner as described above for the primary regulation range in **Next Move Delay (Below and Above) (secs.)**.

Time Step Simulation: Running a Timed Simulation

A [Time Step Simulation](#) can be run either as a Continuous or Timed simulation. The type of simulation is set up on the [Options page](#) in the **Time Step Simulation Options** subsection of the Time Step Simulation dialog.

By default, the **Do Run** button of the [Time Step Simulation dialog](#) will run a **Continuous Simulation**, i.e., a simulation in which each time point is solved immediately after the previous time point. In this case the purpose of the Simulation is to obtain solution for the time points as fast as possible. As the solution progresses, the results for each time point in every time point grid are refreshed, showing the user the evolution of the Simulation. Please read the [Time Step Simulation Quick Start](#) section for a quick introduction on how to run a **Continuous Simulation**.

On the other hand, the Time Step Simulation tool can also be used to run a **Timed Simulation**. In this case, the Simulation takes place according to a time scale proportional to the date/times of the time points. When you click the **Do Run** button, the Simulation progresses as if it was running in actual time. The simulation can also be paused and reset at any time by using the **Pause Solution** and **Reset Run** buttons of the [Time Step Simulation dialog](#).

The following are some of the things you can do with the **Timed Simulation**:

- You can hide the [Time Step Simulation dialog](#) and control the simulation by using the [Time Step Simulation Toolbar](#), which contains buttons such as **Do Run**, **Pause**, **Reset**, etc. that mimic the control buttons of the dialog.
- You can visualize how the quantities vary proportionally to actual time after each time step is applied. Recall that the **Time Scale** defined in the [Options page](#) indicates the relationship between the actual time and the time scale defined for the time points in seconds per hour. Thus, a Time Scale value of 60 will indicate that one hour of time span between two time points will occur in 60 seconds of actual time in the **Timed Simulation**.
- You can also animate the flows of the solution of a time point while you wait for the solution of the next one.
- You can contour the online diagrams and see how the visualization changes as the quantities vary in time, and in addition, you can save those contour diagrams as JPEG or bitmaps for each timepoint.
- Finally, you can act on the system (by closing capacitors, changing generator outputs, etc) before the next time point is applied to simulate operating actions in response to system conditions.

All of these actions are very powerful in presentations on how a power system would evolve in time across different scenarios.

Time Step Simulation: Running OPF and SCOPF Simulations

A [Time Step Simulation](#) can be used to obtain optimal power flow and security-constrained optimal power flow solutions on a time point by time point basis for users that have the OPF and SCOPF add-ons. On the [Summary page](#), the following **Solution Types** can be specified:

- Power Flow
- Unconstrained Optimal Power Flow, which is equivalent to Economic Dispatch
- Optimal Power Flow (OPF)
- Security-Constrained Optimal Power Flow.

Note that different time points can be solved by any of the previously listed solution methods in the same time step simulation. However, the solution settings of the previous time points are used as initial conditions for the solution of the next time point.

Simulator OPF and SCOPF tools are among the most advanced optimization packages for power systems. They have been extended in the latest versions of Simulator with many features, and have become complex analysis system. For users that are unfamiliar with the OPF/SCOPF solutions, read the sections on [Optimal Power Flow](#) and [Security-Constrained Optimal Power Flow](#) before setting up OPF/SCOPF time step simulations.

Power Flow Time Step Simulation

The power flow simulation allows the user to obtain AC or DC power flow solutions for a set of time points. During the Time Step Simulation all the power flow options defined in the [Simulator Options](#) page as well as in other dialogs are used for the solution. A key concept of the Time Step Simulation is that if you select **Solve Time Point** from the [Summary page](#) or from the grids of the [Input page](#), you would obtain the same solution that if you would hit the **Single Solution Button**. This is true, when the input data does not contain schedule data but only time point input data. If your Simulation contains schedule data, there may have been scheduled actions that were applied in previous time points that are not being applied when you select the **Solve Time Point** option.

The power flow solution will observe all the power balance constraints, control limits and area interchange constraint defined in the power flow settings.

Unconstrained OPF Time Step Simulation

In the unconstrained OPF solution, the Time Step Simulation removes all the constraints that would normally act in the OPF and optimizes the system to find the minimum operating cost settings. In doing so, Simulator will change the set points of the specified controls (generators and phase shifters) to minimize the cost of all Areas and Superareas set to OPF AGC control.

Besides the power flow solution options, the Unconstrained OPF simulation will take all the options that have been defined for a regular OPF solution. Most of these options are defined in the [OPF Options Dialog](#), which is accessed using the **OPF Options and Results** button on the [Add Ons](#) ribbon tab. All the settings such as objective function, cost of unenforceable constraints, control available, prices for controls, etc. are defined in this dialog. Other options are defined for each particular object such as generators, loads, areas, lines, interfaces, etc.

OPF Time Step Simulation

When using the OPF solution type, the Time Step Simulation applies the time point and schedule input data and optimizes the control areas set to OPF to minimize cost while enforcing normal operation

constraints: transmission line thermal limits, interface limits, generator control limits, and load control limits. In doing so, the OPF algorithm detects the controls that need to be moved, the constraints that are binding at the solution point, and the unenforceable constraints, i.e., constraints that cannot be enforced with the available controls.

Some of the quantities that are of interest in the solution of the OPF algorithm are:

- **Locational Marginal Prices:** These need to be selected for display on the Buses [Results page](#). Average LMP prices and other LMP metrics can also be displayed in the [Results page](#) for Areas, Injection Groups, Super Areas, and Zones.
- **Binding Constraints** as well as **Marginal Cost of Limit Enforcement** for lines and interfaces. These fields can be seen in the [Results: Constraints page](#) and can be added to the [Results page](#) for Lines, Transformers, and Interfaces.
- **Unconstrained, Final** and **Congestion Cost**, displayed in the **Results Summary Page** for the entire system. These costs can also be displayed for Areas, Owners, Superareas and Zones in the [Results page](#).
- **Profit \$/hr** for Generators, Owners, Superareas, Areas, and Zones can be displayed in the corresponding grids of the [Results page](#).

SCOPF Time Step Simulation

The SCOPF combines the power of Simulator's OPF with the [Contingency Analysis Tool](#) to optimize a system for minimum cost while enforcing both normal operation and contingency constraints. At each time point, the SCOPF solution provides the optimal operation of the system so that if contingencies occur they would not create security violations. The locational marginal prices created in this manner are security-constrained signals to the market.

The solution of SCOPF Time Step Simulation depends on the options that have been set up for the following tools:

- Power Flow
- Optimal Power Flow
- Contingency Analysis
- Security Constrained Optimal Power Flow
- Time Domain OPF Options

The SCOPF Time Step Simulation does the following for each time point:

- Applies the input data to the power system
- Applies scheduled actions determined by the schedule input data
- Solves a power flow
- If specified, solves an unconstrained optimal power flow (economic dispatch)
- Initializes the base case of the security constrained OPF by solving a power flow or an OPF
- For the initial system conditions, solves the list of contingencies
- Solves the SCOPF optimization problem: minimizes operating cost while enforcing normal and contingent constraints
- Displays the results in all the result grids

The SCOPF is on its own a complex computation that often requires significant computer resources. This is due mostly to the solution of the list of contingencies and the calculation of their sensitivities. The size of the problem can be dimensioned by:

- Size of the system, given by the number of buses and the areas to be optimized. This is difficult to reduce since the optimization problem is normally defined for a certain region.
- Number of contingencies, which can be reduced by developing a contingency screening using peak loading conditions.
- Number of constraints (monitored elements), which can be reduced by selecting critical element, e.g. interfaces and higher voltage transmission lines.
- Number of time points in the list.

Another mechanism to speed up the computation of the PF/OPF/SCOPF Time Step Simulation is to use DC solutions in some of the internal routines:

- AC or DC power flow
- AC or DC contingency analysis. This one will produce the larger time savings.
- AC or DC SCOPF

Time Step Simulation: Storing Input Data and Results

In large cases, the amount of data that can be potentially generated by a [Time Step Simulation](#) is significant considering that results for a full PF/OPF/SCOPF solution are available for each time point. A convenient method to store large amounts of data is to save the data in a binary file. The input data, both time point and scheduled data, as well as the results can be stored in a **Time Series Binary File**, referred to in this help guide as the **.tsb file**.

Here is a summary of what will be stored in the **.tsb file** when you click the **Save TSB File** button in the [Time Step Simulation Dialog](#):

- All input data, defined in the [Input pages](#) for MW Load, Mvar Loads, Generator MW, Generator Maximum MW, Line Status, Area Total MW Load, Zone Total MW Load, and Injection Groups.
- All scheduled input data, defined by the combination of Schedules and Schedule Subscriptions specified in the corresponding [Input pages](#).
- The values displayed on the grids of the [Results: Constraints page](#) for Binding Lines, Binding Interfaces, Binding Contingencies, and the Binding Line Matrix and List.
- The customization settings defined on the pages of the [Custom Results Selection Dialog](#) for the time point results of Areas, Buses, Generators, Switched Shunts, Injection Groups, Interfaces, Lines, Loads, Owners, Superareas, Transformers, and Zones.
- The customized results on the pages of the [Results page](#) for time point field values of Areas, Buses, Generators, Switched Shunts, Injection Groups, Interfaces, Lines, Loads, Owners, Superareas, Transformers, and Zones. Recall that many fields can be defined for each type of object.
- The options defined in the [Options page](#), except the options set up in the **Auto Load TSB File Options**, which are saved with the **.pwb case**.
- The data of the last solution run contained in the [Summary page](#): Skip, Processed and Solved fields.
- The **.tsb file** description

Note: Recall that the data of any Simulator grid can be copied to the clipboard and to Excel by selecting the options found on the local menu accessed by right-clicking on the grid.

Fault Analysis

Fault analysis can only be performed when Simulator is in [Run Mode](#). There are three ways to start a fault analysis study:

- Go to the [Tools](#) ribbon tab, and choose **Fault Analysis** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).
- Right click on a bus and choose **Fault...** to perform a fault analysis at that bus
- Right click on a line and choose **Fault...** to perform a fault analysis at that point on the line

All of these options will open the [Fault Analysis](#) dialog. If you opened the dialog by right-clicking on a bus or line, the fault information on that bus or line will already be filled in. If you selected the **Fault Analysis...** option from the [Tools](#) ribbon tab, the information about the location of the fault will need to be provided.

Fault Analysis Dialog

The **Fault Analysis** dialog can be used to perform a fault analysis study on the currently loaded power system. A fault study can only be performed while Simulator is in [run mode](#), since the load flow must be validated and solved before a fault study can be calculated. If you are observing single fault analysis results in the fault analysis dialog and switch to [edit mode](#), the dialog will automatically be closed and the single fault analysis results will be cleared from memory. Multiple fault results displayed in the Fault Definitions table are maintained when switching to edit mode, but may no longer be relevant when switching back to run mode, depending on what was done to the power system data in edit mode.

The fault analysis has two choices for calculating fault results. The first option is calculating the fault current at multiple defined points for a general summary of [several faults](#), and the second is calculating detailed fault results and fault current flows for a [single fault](#) point.

Simulator stores fault data in the PowerWorld binary file along with the load flow data, but by default most other load flow formats store fault data in separate files. The fault data can be stored in and loaded from an external file, but if no fault data is present in a PowerWorld binary file or loaded from an external file before a fault analysis is run, Simulator will use the load flow data as default values for the analysis. Fault data values can also be modified for specific devices by opening a specific device's information dialog and looking at the **Fault...** tab. Devices that require sequence specific data for fault analysis are [buses](#) (for sequence load injections), [generators](#), [switched shunts](#), [transmission lines](#), and [transformers](#).

Phase shifts in a fault analysis calculation can be very important for calculating the correct fault currents and voltages throughout the system. The phase shifts that are applied for transmission lines and transformers are taken from the load flow values of phase entered with each specific transmission element. While transformers can have their transformer configurations specified (i.e. Delta-Wye, Grounded Wye-Delta, etc.), these configurations are **NOT** used to determine phase shift angles, **ONLY** to determine the proper grounding on each side of the transformer. The phase shifts that are applied are taken from the load flow data phase values for the transmission elements. If no phase shifts are entered in the load flow data, the fault analysis will treat all elements as having zero phase shift. Phase shift values can be entered manually for each transmission element, but are also included in most load flow formats and will be read into Simulator when loading a load flow data file.

Note that the bus chosen for the fault is always set to a 0 degree reference, and all other buses are shifted according to this reference.

Load Data / Save Data

These two buttons allow loading from and saving to external files. Currently the two types of files supported are PSS/E Sequence Data files (.seq) and [PowerWorld Simulator Auxiliary files](#) (.aux). Either one of these formats can be loaded and saved.

Note that typically Simulator assumes that if no zero sequence data is given for a branch that the zero sequence impedance is defaulted to 2.5 times the positive sequence impedance. However, in the case of reading in data from a PSS/E sequence data file, you are given the option to instead treat branches with now explicitly given zero sequence data as open circuits in the zero sequence network.

Fault Analysis Bus Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase voltage magnitudes and angles are displayed. In addition, the sequence voltages and angles can also be added by modifying the display using the [Display/Column Options](#) dialog.

Fault Analysis Generator Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the generator. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the generator and into the terminal bus.

Fault Analysis Line Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for each end of the branch. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of or away from a bus.

Mutual Impedance Records

The **Mutual Impedance Records** table is a [Case Information Display](#) and can be customized like any other case information display. The zero sequence mutual impedance records displayed in this table can be either read from a sequence data file, or created manually by choosing **Insert...** from the local menu.

The common fields displayed on the Mutual Impedance Records display are:

L1 From Bus, L1 To Bus, and L1 Ckt ID

These fields represent the from bus number, to bus number, and circuit identifier for the first mutually coupled line.

L2 From Bus, L2 To Bus, and L2 Ckt ID

These fields represent the from bus number, to bus number, and circuit identifier for the second mutually coupled line.

Mutual R, Mutual X

The mutual impedance, in terms of the resistance and reactance (per unit). The dot convention of the mutual impedance assumes the From bus of each line to be the dotted terminal, with the sign of the mutual impedance values being set according to this convention.

L1 Mut. Start, L1 Mut. End

The starting point and ending point of the mutually coupled portion of the first mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

L2 Mut. Start, L2 Mut. End

The starting point and ending point of the mutually coupled portion of the second mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

Mutual Impedance Record Dialog

The **Mutual Impedance Record** dialog can be used to modify or add zero sequence mutual impedance records to the sequence data for a case. When the dialog is opened using the **Show Dialog...** option from the [Mutual Impedance Records](#) table local menu, the information for the record selected in the table will automatically be displayed. The information for that record can be modified, or a different record can be selected by selecting different lines in the Line 1 and Line 2 Identifier sections. Note that the drop down list of buses for the From Bus fields always contain all the buses in the case. However, once the From Bus has been selected, the drop down list of the corresponding To Bus field will only contain bus numbers of buses that are connected to the From Bus. If a mutual impedance record already exists for the lines selected, the information for that record will be displayed. If a mutual impedance record does not exist for the selected lines, then the mutual impedance fields will display default values. When the default values are changed, and either Save or OK are selected, a new mutual impedance record is added to the data.

Fault Analysis Load Records

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the load. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the load.

Fault Analysis Switched Shunt Records

This display has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the switched shunt. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the switched shunt.

PowerWorld Simulator PV/QV Overview

The PV and QV tools are only available if you have purchased the PVQV add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the PVQV version of Simulator.

PVQV, PowerWorld's voltage adequacy and stability assessment add-on, is used to analyze the voltage characteristics of a power system.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPF](#)) among other add-ons. For more information see .

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the base package, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However, with the voltage adequacy and stability (**PVQV**) add-on the user can solve multiple power flow solutions in order to generate a [PV curve](#) for a particular transfer or a [QV curve](#) at a given bus.

The PVQV functionality is accessed using the PV and QV Curves (PVQV) ribbon group from the [Add Ons ribbon tab](#). The commands available in this ribbon group are [Refine Model](#), [PV Curves](#), and [QV Curves](#).

The purpose of the PVQV add-on is monitor system voltages, or other user specified parameters, as a real power transfer is increased or reactive power is injected at selected buses. The PVQV add-on uses the Simulator built-in Newton Raphson power flow algorithm to accomplish this task. After PV and QV simulations are completed, the user can choose to graph any of the monitored system parameters, designated in [Quantities to Track](#).

PV Curves

Simulator's **PV Curves** tool provides the ability to produce plots of maximum power transfer (PV) curves for voltage at any bus in the system, as well as 2-dimensional plots of various quantities tracked during the simulation. In Run Mode, select **PV Curves...** from the **Add Ons** ribbon tab to open the [PV Curves dialog](#). The PV Curves dialog allows you to specify the quantities to be tracked during the simulation, control solution parameters, set violation boundaries, and run the PV analysis.

The PV curve process will automatically solve a sequence of power flows at incremental levels of power transfer between a source Injection Group and a sink Injection Group. [Injection Groups](#) may contain generators, loads, or a combination of both. Each **Contingency** defined in the [Contingency Analysis Tool](#) that is not flagged to be skipped will be processed and tracked as a separate **PV Scenario**, along with the pre-contingent base case, at each incremental power transfer. After each successful power flow solution, the user-specified [Quantities to Track](#) are recorded and the transfer is increased by the next increment. If a power flow solution fails, then the last successful solution for that scenario is applied and a smaller incremental transfer is applied. The process concludes after finding the maximum power transfer for the user-specified number of critical scenarios.

PV Curves Dialog

To display this dialog, go to the [Add Ons](#) ribbon tab and select **PV Curves** from the **PV and QV Curves (PVQV)** ribbon group.

The PV Curves dialog contains all of the setup and controls for processing and analyzing the PV curve analysis. The dialog is broken down into several pages:

[Setup](#)

[Quantities to track](#)

[Limit violations](#)

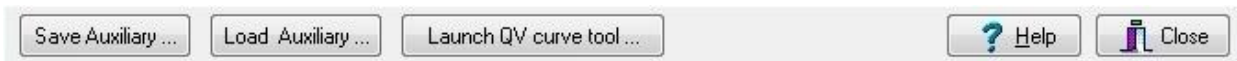
[PV output](#)

[QV setup](#)

[PV results](#) (Note: Launching the analysis is done from this page.)

[New Plots](#)

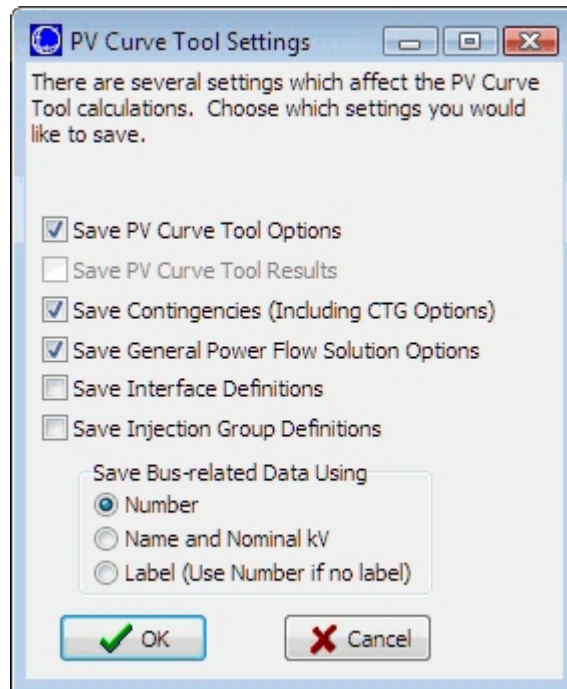
There are several buttons at the bottom of the dialog that are available regardless of the selected page.



Save Auxiliary...

Clicking this button will prompt for a filename in which to save an [auxiliary file](#). PV-specific results and option settings are not saved with the case when it is saved to file. Use this option to store any results and settings that need to be retained for future use.

The PV Curve Tool Settings dialog shown below will be displayed with options for specific data to include in the auxiliary file. Check the box next to a particular data set to save this data in the auxiliary file. Which key field to use when identifying objects in the file can also be specified on this dialog. Click **OK** on this dialog to finalize saving the file or **Cancel** to abandon the file save.

**Load Auxiliary...**

Clicking this button will open a dialog to allow the user to select an [auxiliary file](#) to load. This is intended to be used for loading any relevant option settings to be used during the PV analysis. The dialog will be updated according to the option settings loaded from the file.

Launch QV curve tool...

This button will launch the [QV Curves](#) tool using the options specified on the [QV setup](#) tab. This button is disabled when using [Integrated Topology Processing](#).

Help

Displays this help page.

Close

This will close the dialog without running the analysis. The currently set options will be saved, and the dialog will not close unless all option settings are valid.

PV Curves: Setup

The Setup tab is found on the [PV Curves dialog](#).



Setup

Transfer power between the following two groups:

Source

Sink

View / Define Groups

Common Options Advanced Options

The Setup tab contains the options for defining the source and sink for the PV study transfer. The PV tool expects the source and the sink to be [injection groups](#) defined by the user.

Source

Use this drop-down box to identify the source [injection group](#). To model an increase in transfer, generator points in the source injection group will increase their output, and load points will decrease their magnitude in amounts proportional to their participation factors.

If the injection group to use for the source has not already been created, click the View/Define Groups button in order to create it. After the injection group has been created, it will show up in this drop-down box and can then be selected.

Sink

Use this drop-down box to identify the sink [injection group](#). To model an increase in transfer, generator points in the sink injection group will decrease their output, and load points will increase their magnitude in amounts proportional to their participation factors.

If the injection group to use for the sink has not already been created, click the View/Define Groups button in order to create it. After the injection group has been created, it will show up in this drop-down box and can then be selected.

View/Define Groups

If injection groups have not been previously defined in the current case, they can be created by clicking on this button. To create a new injection group, right-click on the resulting injection group [case information display](#) and choose **Insert** from the popup menu.

Alternatively, if a list of injection groups has been previously saved in an [auxiliary](#) file, they can be loaded into the current case by right-clicking on the resulting injection group case information display and choosing **Load > Auxiliary File** from the popup menu.

Once the source and sink points have been defined, the PV study will model an increasing transfer of power from the source points to the sink points. The transfer process is performed incrementally, based upon user specified [options](#) on how the transfer should vary during the solution process.

The remainder of the setup options are found on sub-tabs for [Common Options](#) and [Advanced Options](#).

PV Curves Setup: Common Options

These options are located the Common Options sub-tab found on the [Setup](#) tab of the [PV Curves dialog](#). These are the options most commonly adjusted for the PV analysis.

The screenshot shows the 'Common Options' sub-tab of the 'PV Curves Setup' dialog. The 'Advanced Options' sub-tab is also visible. The 'Vary the transfer as follows:' section contains several settings: 'Initial Step Size (MW)' is 100, 'Minimum Step Size (MW)' is 10, 'When convergence fails, reduce step by a factor of' is 2, 'Stop when transfer exceeds' is 0, and 'Island-Based AGC Tolerance' is 5.0000. There are three checkboxes: 'Allow only AGC units to vary' (unchecked), 'Enforce unit MW limits' (unchecked), and 'Do not allow negative loads' (unchecked). The 'Dispatch generators in merit order:' section has two checkboxes: 'Source' (unchecked) and 'Sink' (unchecked). The 'Critical Scenarios' section has a label 'Stop after finding at least' followed by a spinner set to 1 and the text 'critical scenarios'. The 'Base Case and Contingencies' section has two checkboxes: 'Skip contingencies' (unchecked) and 'Run base case to completion' (unchecked). There are two buttons: 'Manage contingency list ...' and 'Base Case Solution Options ...'.

Initial Step Size (MW)

This option indicates the initial increment in which the transfer will be increased following each successful iteration. The default value is 100 MW.

Minimum Step Size (MW)

Whenever the PV process fails to solve the system at a given transfer level, it will return to the previously solved transfer level, reduce the step size by the specified factor, and then try to solve the system with the transfer incremented by the newly reduced step size. The **Minimum Step Size** option specifies the minimum size this increment can be. Once the system fails to solve when the step size is at this value, the process will conclude that we have come very close to the voltage collapse point and terminate the analysis. So, the minimum step size essentially functions as a tolerance for computing the voltage collapse point. The default value is 10 MW. The minimum step size must be greater than or equal to 0.1 MW.

See the topic below about **Tolerances in PV Tool** for more information about how the minimum step size interacts with the Island-Based AGC Tolerance and power flow MVA Convergence Tolerance.

When convergence fails, reduce step by a factor of...

Whenever the PV process fails to solve the system at a given transfer level, it will reduce the transfer step size by the value specified for this option. The default value is 2. Therefore, the process will start incrementing the transfer in 100 MW steps. When it reaches a transfer level that it cannot solve, it will return to the last solved transfer level, reduce the step size to 50 MW, increment the transfer by 50 MW, and attempt to solve the case again. The next time it fails to solve, it will reduce the step size to 25 MW, and then to 12.5 MW, and finally to 6.25 MW. Since 6.25 MW is less than the **Minimum Step Size** value of 10 MW, it will instead use a final step size of 10 MW. Once the system fails to converge with this step size, the analysis will terminate because it concludes that

it has arrived at the voltage collapse point, within the specified tolerance. This value must be greater than or equal to 1.01.

Stop when transfer exceeds

Check this box to terminate the PV analysis once the transfer between the source and sink reaches an amount equal to the MW value specified.

Island-Based AGC Tolerance

When implementing the transfer, island-based AGC is used. Injection changes for the transfer are first made to the source injection group, and when the power flow is solved, the sink injection group is adjusted for any injection changes due to the source injection group changes and any changes in losses due to the transfer. Because the process to determine required injection changes during the power flow solution MW control loop is an iterative process, a tolerance value must be specified to determine when the changes to the sink injection group are acceptable. This tolerance is NOT the same value that is set with the [power flow solution options for specifying island-based AGC](#).

As a general rule of thumb, the Minimum Step Size should be at least 2 times larger than this tolerance. If not, all changes due to an incremental transfer that is smaller than this tolerance will actually occur between the source and the system slack instead of the source and the sink.

See the topic below about **Tolerances in PV Tool** for more information about how the AGC tolerance interacts with the **Minimum Step Size** and power flow **MVA Convergence Tolerance**.

Allow only AGC units to vary

Individual generating units are distinguished according to whether they do or do not participate in the automatic generation control (AGC) effort. Each generator has a field that determines whether or not that particular generator is AGC-able. By checking this option, only those generators whose AGC field is set to *YES* will contribute to the power transfer being studied. Otherwise, all units in the injection groups will be allowed to participate regardless of their AGC status.

Enforce unit MW limits

If this option is checked, the output of any participating generating unit will be kept within its designed operating range of $\text{MinMW} < \text{Output} < \text{MaxMW}$. When a unit is pegged at one of its limits, participation factors of the other points in the limited generator's injection group will be adjusted to pick up the difference.

If this option is checked and during the ramping all units in either the source or sink hit their limits, the scenario for which the ramping is being done will be considered critical. The [Critical Reason](#) for the scenario will reflect whether the source or sink ran out of resources.

Do not allow negative loads

This is the analog of the previous option for loads. If a load is used as a source point (or sink point when choosing to apply the [reverse transfer](#)), it will be decreased to make power available for the transfer. Checking this option will instruct the PV process to keep loads from falling below 0 MW. If a particular load is capped at 0 MW, participation factors for the remaining points in its injection group will be recalculated to make up the difference. If there are not enough resources in the injection group to continue with the transfer, the scenario for which the ramping is being done will be considered critical. The [Critical Reason](#) for the scenario will reflect whether the source or sink ran out of resources.

Dispatch generators in merit order

Two different methods of merit order dispatch are available. The basic method is implemented by checking the **Use** box for the **Source** or **Sink**. When using the basic method, injection group generators will be dispatched by moving individual generators to their maximum/minimum outputs in succession based on their relative participation factors ordered from highest to lowest. To

dispatch the source injection group in merit order, check the box next to **Source**. To dispatch the sink injection group in merit order, check the box next to **Sink**. When using the basic option, generator unit MW limits will be enforced for those units participating in the merit order dispatch regardless of how the **Enforce unit MW limits** option is set.

The second method involves dispatching generators so that they are dispatched within an economic range. To implement this option, the **Use** box must be checked along with the **Use Economic** box. To dispatch the source injection group in economic merit order check the boxes next to **Source**, and to dispatch the sink injection group in economic merit order check the boxes next to **Sink**. Details about how economic merit order dispatch is performed can be found under the [Generator Economic Merit Order Dispatch](#) topic.

When using either of the merit order options and an injection group contains loads along with generators, only the generators will be adjusted and the loads will not participate in the injection changes. If using merit order dispatch and an injection group contains ONLY loads, the loads will be adjusted according to their participation factors and merit order dispatch will not be used.

Stop after finding at least ... critical scenarios

A large number of contingencies can be specified for analysis. Usually, there is only concern with determining a certain number of the most critical scenarios instead of running each scenario to its critical point. Set this number to stop the analysis of all scenarios once the specified number of critical scenarios has been found. It should be noted that this number really serves to find *at least* the specified number of critical scenarios. Depending on when a scenario becomes critical in the process, it is possible that more than the specified number of critical scenarios will be found.

Before implementing a new transfer step for all contingencies, the number of scenarios that are already critical is checked against the number of critical scenarios to find. If at that point the number found is greater than or equal to the number specified, the process will stop. At each transfer step, all contingencies that are not already critical are implemented and a power flow solution is attempted for each contingency. If any contingency does not solve, that contingency is evaluated independently to determine the transfer level at which it becomes critical. If multiple contingencies cannot be solved at a particular transfer step, all of these contingencies will be evaluated independently to determine the transfer level at which it becomes critical. All contingencies are evaluated even if this would cause the number of critical scenarios found to exceed the specified number of critical scenarios to find.

Skip contingencies

The PV Curve tool computes PV curves for both the base topology and for any contingencies that have been defined, unless the **Skip contingencies** checkbox is checked. If the **Skip contingencies** checkbox is checked, a PV curve will be computed only for the model in its present (base case) topology.

Manage contingency list...

Clicking this button will open the [contingency analysis dialog](#) for managing or inserting contingencies to be processed during the PV analysis. Contingencies defined and marked for processing in the contingency analysis dialog will be included. Only those contingencies whose **Skip** field is set to *NO* will be included in the PV analysis.

Note: The Robust Solution Process option that can be utilized with the contingency analysis when a contingency solution failure occurs is not applied when a contingency is implemented during the PV analysis. Most other options defined with contingency analysis such as contingency specific solution options and make-up power specifications are used when applying a contingency during the PV analysis. The exception here is for the contingency-specific solution option for Minimum Per Unit Voltage for Constant Power Load. This is always set to 0 pu during the PV analysis.

Run base case to completion

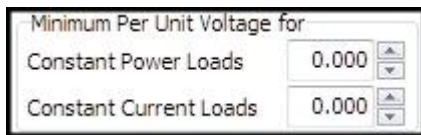
The PV curve tool is designed to ramp a transfer until the prescribed number of critical scenarios, including both critical contingencies and critical base topology, have been found. If the requested number of critical scenarios have all been identified as being associated with contingencies, the tool will not reveal how much a transfer can be ramped for the base topology, unless the **Run base case to completion** checkbox is checked. Checking this checkbox forces the tool to continue to ramp the transfer until the base case can no longer be solved, regardless of whether the requested number of critical transfer level / topology combinations have been found.

If using the **Stop when transfer exceeds** option, it is possible that the base case will not be run to completion if the specified transfer level is exceeded before the critical point for the base case is found.

Base Case Solution Options ...

Click this button to bring up the [Simulator Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affect all power flow computations, even those outside the PV process.

Because the goal is to stress the system while performing the PV analysis, there are two power flow solution options that are set internally as part of the PV process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.



These options are both set to 0 pu during both the base case solution and any contingency case solution.

Tolerances in PV Tool

Tolerances must be set correctly with the PV tool or injection changes will end up being picked up by the system slack instead of the sink injection group or no injection changes will be made at all. The **Minimum Step Size** will dictate what the **MVA Convergence Tolerance**, set with the [Power Flow Solution Options](#) on the [Simulator Options Dialog](#), and the **Island-Based AGC Tolerance** can be. Before the analysis starts, the MVA Convergence Tolerance will be checked to make sure that it is less than $0.1 * (\text{Minimum Step Size})$. If not, the MVA Convergence Tolerance will be set to $0.1 * (\text{Minimum Step Size})$. The Island-Based AGC Tolerance will also be checked to make sure that it is less than $0.5 * (\text{Minimum Step Size})$. If not, the Island-Based AGC Tolerance will be set to $0.5 * (\text{Minimum Step Size})$. The MVA Convergence Tolerance will then be checked to make sure that it is less than $0.2 * (\text{Island-Based AGC Tolerance})$. If not, the MVA Convergence Tolerance will be set to $0.2 * (\text{Island-Based AGC Tolerance})$. This will order the tolerances such that $(\text{MVA Convergence Tolerance}) < (\text{Island-Based AGC Tolerance}) < (\text{Minimum Step Size})$. The original tolerances will be restored when the initial state stored with the PV tool is restored.

Another issue dealing with tolerances is cumulative system slack error. The cumulative system slack error will be recorded after each ramping step. This is done so that an attempt can be made to keep the system slack from deviating more than the Island-Based AGC Tolerance over all of the ramping by adjusting for this error during the next ramping step. This error will be applied to the amount of change required from the sink injection group in an attempt to bring the system slack back towards its original output. Without this accounting it is possible that the system slack could move by the Island-Based AGC Tolerance at each ramping step, and the overall results would be that the system slack could move $(\text{Island-Based AGC Tolerance}) * (\text{Number of ramping steps})$ by the completion of the process. With this accounting, the system slack may move up and down slightly over the course

of all ramping, but on average will stay near its starting output within the Island-Based AGC Tolerance.

PV Curves Setup: Advanced Options

These options are found on the Advanced Options sub-tab located on the [Setup](#) tab of the [PV Curves dialog](#).

Common Options **Advanced Options**

How should reactive power load change during ramping?

☐ Maintain the MW/MVAR ratio at each load, but then scale MVAR by a factor of 1.0000

☒ As MW changes, change the MVAR at a power factor of 1.0000

Load Component Variation

☒ All changes apply to constant power (S MW, S MVAR)

☐ Vary in proportion to existing Z,I,P ratios

☐ Vary using proportions specified below:

	P (MW)		Q (MVAR)	
	Source	Sink	Source	Sink
Power (S)	1.00	1.00	1.00	1.00
Current (I)	0.00	0.00	0.00	0.00
Impedance (Z)	0.00	0.00	0.00	0.00

Reverse Transfer

☐ Apply Reverse Transfer

Maximum Reverse Transfer (MW > 0) 1000

The power transfer that occurs during the PV analysis is a real power (MW) transfer. Loads can be specified as part of either the source or sink injection group that define the transfer. When loads are included in the transfer, the default is to only adjust the real power (MW) component of the load and leave the reactive power (Mvar) component constant. If any change is made to the reactive power load (Mvar) during the transfer, the default is to have all changes be made to the constant power component of the load. The Advanced Options tab allows specification of load variation for both real and reactive load to be something other than the defaults during the transfer.

The transfer amount and how the transfer is adjusted are determined by options found on the [Common Options](#) tab.

How should reactive power load change as real power load is ramped?

Total read power load changes, ΔP , at each load are determined based on the load's participation factor as defined with its associated injection group and the amount of transfer that is being implemented. ΔP is then used in conjunction with the selected reactive power change option to determine the associated total reactive power change, ΔQ , at each load.

Maintain the MW/MVAR ratio at each load, but then scale MVAR by a factor of

The total reactive power change, ΔQ , is determined based on the power factor, pf , at each nominal load (P_{nom}, Q_{nom}) prior to the load change and the change in total real power load, ΔP , due to the transfer. The power factor is determined by the following:

$$pf = \cos \left(\tan^{-1} \left(\frac{Q_{nom}}{P_{nom}} \right) \right)$$

An additional multiplier, $pfQMult$, can be specified in order to modify the total reactive power load change to allow adjustment away from the present power factor. This multiplier is 1.0 by default. The change in reactive power load is then:

$$\Delta Q = \tan \left(\cos^{-1} (pf) \right) * \Delta P * pfQMult$$

Maintaining a constant MW/MVAR ratio implies a sign convention that is lost when applying the arctan function to the ratio. An additional check is done to make sure that the calculated ΔQ is in the correct direction (maintains the correct leading or lagging power factor).

$$\text{if } \frac{Q_{nom}}{P_{nom}} < 0 \text{ then } \Delta Q = -1 * \Delta Q$$

$$\text{if } P_{nom} = 0 \text{ then } \Delta Q = 0$$

When loads are specified as constant power, constant current, and constant impedance components (ZIP components) rather than simply all constant power, the total load becomes a function of voltage. Maintaining a constant power factor at each load requires that we use nominal load (load at 1.0 pu voltage) that does not change as a function of voltage. When only constant power components are specified, the actual load is the same as the nominal load. Therefore, in all situations the power factor is determined based on the nominal load.

As MW changes, change the MVAR at a power factor of

This option allows the specification of power factor, $pf_{specified}$, for defining the total reactive power change, ΔQ , at a load. The change in reactive power load is then dependent on the change in total real power load due to the transfer, ΔP , and this specified power factor.

$$\Delta Q = \tan \left(\cos^{-1} (pf_{specified}) \right) * \Delta P$$

For this option, ΔQ is assumed to be in the same direction (has the same sign) as ΔP .

Load Component Variation

The total real power load change, ΔP , at a load during a PV transfer is determined by the load's participation factor as defined with its associated injection group and the amount of transfer that is being implemented. The total reactive power load change, ΔQ , is determined by the option selected for how reactive power should change during ramping as explained in the previous section. Total load change, $(\Delta P, \Delta Q)$, at a load can be broken into ZIP components for constant power $(\Delta P_s, \Delta Q_s)$, constant current $(\Delta P_i, \Delta Q_i)$, and constant impedance $(\Delta P_z, \Delta Q_z)$. During the PV transfer, the user can specify how changes in load due to the transfer should be split among these components.

The constant current and constant impedance components are a function of voltage and the nominal values specified for these components. Because of this, it becomes important to determine the load changes for the components based on nominal voltage and then apply these changes to the nominal components.

All changes apply to constant power (S MW, S MVAR)

All changes to load will be made to the constant power component:

$$\Delta P_s = \Delta P, \Delta P_i = 0, \Delta P_z = 0, \Delta Q_s = \Delta Q, \Delta Q_i = 0, \Delta Q_z = 0$$

The constant power component of load is not dependent on voltage so the total load change can be applied directly to the constant power component.

Vary in proportion to existing Z,I,P ratios

This option will change the load at a given load such that the ratios of the ZIP components do not change during the load adjustment. The existing ZIP ratios at a load are determined based on the present nominal load prior to any change due to the transfer. The existing total nominal power (P_{nom}, Q_{nom}) at a load is the sum of the components:

$$P_{nom} = P_{nomS} + P_{nomI} + P_{nomZ}$$

$$Q_{nom} = Q_{nomS} + Q_{nomI} + Q_{nomZ}$$

The component ratios are simply the ratio of each nominal component to the total nominal load at a load:

$$k_{PS} = \frac{P_{nomS}}{P_{nom}}, k_{PI} = \frac{P_{nomI}}{P_{nom}}, k_{PZ} = \frac{P_{nomZ}}{P_{nom}} \quad \text{if } P_{nom} \neq 0$$

$$k_{QS} = \frac{Q_{nomS}}{Q_{nom}}, k_{QI} = \frac{Q_{nomI}}{Q_{nom}}, k_{QZ} = \frac{Q_{nomZ}}{Q_{nom}} \quad \text{if } Q_{nom} \neq 0$$

$$k_{PS} = 1, k_{PI} = 0, k_{PZ} = 0 \quad \text{if } P_{nom} = 0$$

$$k_{QS} = 1, k_{QI} = 0, k_{QZ} = 0 \quad \text{if } Q_{nom} = 0$$

The resulting change in nominal load for each load component at a load is then the product of the ratio of each component and the total nominal power change:

$$\Delta P_{nomS} = k_{PS} * \Delta P_{nom}$$

$$\Delta P_{nomI} = k_{PI} * \Delta P_{nom}$$

$$\Delta P_{nomZ} = k_{PZ} * \Delta P_{nom}$$

$$\Delta Q_{nomS} = k_{QS} * \Delta Q_{nom}$$

$$\Delta Q_{nomI} = k_{QI} * \Delta Q_{nom}$$

$$\Delta Q_{nomZ} = k_{QZ} * \Delta Q_{nom}$$

Where the total nominal power change is determined from the total power change required at a load as a function of the present voltage, V , at the bus:

$$\Delta P_{nom} = \frac{\Delta P}{k_{PS} + k_{PI} * V + k_{PZ} * V^2}$$

ΔQ_{nom} is calculated from ΔP_{nom} based on the selection of the option on how reactive power should change during the ramping as described in the previous section.

Vary using proportions specified below:

The factors are divided into four groups: real power load in the source injection group, real power load in the sink injection group, reactive power load in the source injection group, and reactive power load in the sink injection group. Each of these groups determines how the various components will change based on whether the load change is being done to real or reactive load and whether the load is part of the source or sink injection group. The default setting for each group is to apply changes only to the constant power component; **Power (S)** factor is 1 while all other factors are 0. The sum of the factors for each grouping must add up to 1. The

Impedance (Z) factors are not enterable and are calculated to ensure that the sum is always 1. Simply adjust the **Power (S)** and **Current (I)** factors to appropriate values and the **Impedance (Z)** factor will be set automatically.

The calculation for determining the change in load components is done the same as described above when using existing ZIP ratios except that the ratios are user-specified here.

Reverse Transfer

The standard way of performing a PV analysis is to increase a transfer from the source to the sink in positive step increments determined by the Initial Step Size and other parameters that control the size of the step. When a contingency does not solve at zero transfer (base case conditions), it might be possible to find a solvable point by adjusting generation and/or load in the system. The Reverse Transfer option will attempt to find this solvable point by increasing a transfer from the sink to the source (instead of source to sink). This is done by incrementing the transfer from the source to the sink in negative step increments, effectively making the transfer go from the sink to the source.

When the **Apply Reverse Transfer** box is checked, an attempt will be made at finding a solvable point by applying the reverse transfer (sink to source) for any contingency that will not solve at zero transfer. A condition for this option to be used is that the base case power flow must solve.

When choosing to attempt the reverse transfer for contingencies that will not solve, all unsolvable contingencies will be processed even if the number of unsolvable contingencies exceeds the number of critical scenarios specified on the [Results](#) tab. The reverse transfer process will be done for all unsolvable contingencies prior to the standard PV process. After the reverse transfer process completes, the standard PV process (positive transfer from source to sink) will be performed on all contingencies that do solve at zero transfer level as long as the number of critical scenarios specified is not exceeded already by the number of contingencies that wouldn't solve as zero transfer.

The initial step magnitude by which the transfer is increased is the Initial Step Size specified with the Common Options. When a transfer level is found at which a contingency will solve, the step magnitude is reduced by the reduction factor specified with the Common Options. This is done in an attempt to find the minimum transfer required to reach a solvable point. This process will continue until a solvable point is found for each contingency of the **Maximum Reverse Transfer (MW > 0)** threshold is exceeded. It is likely that no solvable point can be found for some contingencies. The Maximum Reverse Transfer threshold should be set such that a reasonable transfer amount is attempted before abandoning the attempt at finding a solvable point. This process could be very time consuming if there are a number of contingencies that do not solve at zero transfer.

In addition to attempting to find a solvable point, the reverse transfer process can also attempt to find a solvable point at which all voltages are considered adequate. To check for adequate voltages during the reverse transfer process, check the option to **Stop when voltage becomes inadequate** that is found on the [Limit Violations](#) tab of the PV Curves dialog. This will force the process to find a reverse transfer point at which the contingency solves and all voltages are above the voltage level specified as inadequate. The inadequate voltage level is specified with the **Voltage level to consider inadequate** option also found on the Limit Violations tab.

If a reverse transfer point is found at which a contingency will solve, and voltages are adequate if choosing to include that check, the [results](#) for the scenario will list the **Critical Reason** as *SRT - original critical reason* with *original critical reason* being the reason why the contingency was considered not to be solved at the zero transfer level. The **Max Export**, **Max Import**, and **Max Shift** values that are reported are the minimum (in magnitude) transfer levels at which the contingency will solve and any voltage conditions are met. Values for any parameters that are being tracked through [Quantities to Track](#) will also be recorded at this transfer level. If there are multiple steps at which the contingency will solve and any voltage conditions are met while trying to find the minimum step, multiple transfer level points will be recorded making it possible to plot a few points of a PV curve.

If during the reverse transfer process no point can be found for which a contingency will solve and voltage conditions are met, the original results will be reported as if the reverse transfer process was not attempted.

If enforcing generator MW limits and/or not allowing negative loads during the transfer causes either the source or sink to not have enough resources to meet the required transfer, two other Critical Reason messages are possible. If the sink is maxed during the transfer, the **Critical Reason** will be given as *RT Sink Maxed - original critical reason*. If the source is maxed during the transfer, the Critical Reason will be given as *RT Source Maxed - original critical reason*. The **Max Export**, **Max Import**, and **Max Shift** values reported will be the values at which either the source or sink hit limits. Either of these messages indicates that the contingency will not solve or voltage conditions are not met, but the reverse transfer cannot continue because there are not enough resources in either the source or sink to do so. If either the source or sink is at its limits the contingency solution will still be attempted. If the contingency does solve and the voltage conditions are met, the **Critical Reason** will be given as *SRT - original critical reason*.

When the reverse transfer process completes, the system state is returned to the base case state in place when the PV analysis was first initiated. No reverse transfer amount will remain in the system state upon completion of the reverse transfer process even if the analysis is such that only the reverse transfer process is completed and no forward transfer scenarios are attempted.

PV/QV Quantities to Track

Specification of quantities to track during analysis is almost identical for both the PV and QV analysis (exceptions will be noted below). For PV analysis, the Quantities to Track tab is accessed from the [PV Curves](#) dialog, and for the QV analysis, this tab is accessed from the [QV Curves](#) dialog. Quantities to track are monitored and stored as the transfer increases for the PV analysis or the voltage set-point changes for the QV analysis. Any unselected system parameters will not be saved. The Quantities to Track page contains several sub-tabs that allow the monitoring of different types of objects including: buses, generators, injection groups, lines, transformers, shunts, interfaces, and the case as a whole. There is also a sub-tab for monitoring devices at limits.

Each of the object sub-tabs contains fields that can be tracked during the PV analysis. The default tracking for all of these fields is *NO*. To monitor a particular quantity, double-click the corresponding field entry to toggle it to *YES*, or in the case of some line quantities, the appropriate monitoring direction.

Quantities that are tracked can be plotted using the [Plot](#) page. They can also be stored to an auxiliary file with both the PV and QV tools. With the QV analysis a special file is created that stores the tracked quantities. For more information on this see the Extra Monitoring File section with the [QV Curves Options: Output](#) topic.

Buses

Voltage - per unit voltage

kV Voltage - actual voltage in kV

Angle - voltage angle in degrees

MW Load - total real power load

Mvar Load - total reactive power load

dV/dQ - sensitivity of voltage to change in reactive power injection at the same bus

VP Sensitivity - sensitivity of voltage change due to real power transfer. For PV analysis this is the sensitivity of the voltage at the bus due to the selected transfer from source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

VP sensitivities can also be calculated using the [Flow and Voltage Sensitivities tool](#). With this tool there is an option to turn AVR control off for generators and continuous switched shunts that are at buses participating in the transfer. This same option is not available when calculating these sensitivities during the PV analysis because of the overhead of additional power flow solutions that would be required at each transfer step to implement this option. Leaving devices on AVR control will maintain the voltage setpoint at the regulated buses creating PV buses in the power flow Jacobian. At PV buses the sensitivity of voltage to a transfer will be zero.

Shunt Inj - total switched shunt reactive power (Mvar) output

For PV analysis if the intent is to trace PV curves, it makes sense to monitor at least one bus voltage. The process will not automatically monitor and store voltage quantities.

For QV analysis, per unit voltage will be monitored automatically for all buses selected to be included in the QV analysis.

If the [Integrated Topology Processing](#) add-on is available, an additional option will be available on the Buses tracking tab, the **Modify Existing Bus Tracking to Track Only Single Bus Per Super Bus** button. When using topology processing, only a single bus per super bus will be in the consolidated case. That means that the parameters for each bus in the super bus will be the same, i.e. same voltage magnitude and angle, etc. This button can be used to eliminate any excess

tracking that would just take up computer memory by tracking essentially the same quantities at the same bus multiple times. To take advantage of this option and only track a single bus per super bus, first set up the bus tracking to track all of the necessary quantities. Then push the **Modify...** button. The field entries in the table will then be updated to only include a single bus per super bus. When determining the primary node (the primary node is the bus that will ultimately be tracked during the analysis) and super buses, the active defined contingencies will be used to determine how the case is consolidated. Because the consolidation can be different based on the contingency set, or lack thereof, it is best to just define the tracking to include all necessary buses and use this option to refine the list down to a single bus per super bus rather than manually trying to maintain such a list.

Generators

Gen MW - real power output

Gen Mvar - reactive power output

Mvar Reserve - reactive power reserves in the positive direction (Max Mvar - Mvar)

Groups

This allows the monitoring of injection group parameters.

Mvar Reserve - total reactive power reserves in the positive direction for all generators in the group (Max Mvar - Mvar)

Gen MW - total real power output of all generators in the group

Gen Mvar - total reactive power output of all generators in the group

Load MW - total real power load for all loads in the group

Load Mvar - total reactive power load for all loads in the group

Lines

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab.

Branch MW - real power flow on the branch in either the FROM-TO or TO-FROM direction

Branch Mvar - reactive power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MVA - total power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MW Loss - real power loss

Branch Mvar Loss - reactive power loss

Branch PTDF - [Power Transfer Distribution Factor](#) on the branch due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Xfmr Tap - tap ratio if the branch is a transformer

For the flow (Branch MW, Branch Mvar, and Branch MVA) and Branch PTDF fields, double-clicking on a particular entry will toggle its value from *NO* to *FROM-TO*, and double-clicking again will toggle its value to *TO-FROM*. Double-clicking on a loss field or Xfmr Tap field will toggle the value between *YES* and *NO*.

Transformers

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. See **Lines** above for setting up common branch quantities. Only branches containing transformers will appear on the **Transformers** sub-tab.

Type - set the transformer type (Fixed, LTC, Mvar, or Phase)

Reg Val - regulated value for the transformer control

Tap Pos - tap position

Reg Err - deviation of the regulated value from the regulation Min/Max range

All fields specified above except for the Type field are used to determine which quantities should be monitored and stored during the analysis. The Type field can be accessed here in order to change the type of transformer. The type will not be monitored during the analysis.

Shunts

This allows the monitoring of switched shunt parameters.

Actual Q - actual reactive power output

Actual P - actual real power output

Nom Q - nominal reactive power output

Nom P - nominal real power output

Reg Err - deviation of the regulated value from the regulation Min/Max range. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Reg Val - value of regulated quantity at the regulated bus. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Interfaces

MW Flow - real power flow

Mvar Flow - reactive power flow

MVA Flow - total power flow

MW Loss - real power losses

Mvar Loss - reactive power losses

Interface PTDF - [Power Transfer Distribution Factor](#) on the interface due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Case

These options allow the tracking of case quantities to provide a better picture of what is going on in the case as a whole without requiring the individual tracking of quantities that make up the case total. These quantities are determined in the same manner as those found in the Case Totals section of the [Case Summary dialog](#).

Available quantities for tracking include:

Load MW/Mvar - total real or reactive power load for the entire case

Generation MW/Mvar - total real or reactive power generation for the entire case

Shunts MW/Mvar - total real or reactive shunt injection from switched shunts, bus shunts, and line shunts. A positive value will indicate shunt load where a negative value will indicate shunt compensation such as capacitance.

Losses MW/Mvar - total real or reactive power losses for the entire case

ATC Extra Monitors

This option is only available when using the PV Curves tool. The following will appear at the bottom of the Quantities to Track tab accessed from the PV Curves dialog.



[ATC Extra Monitors](#) are typically used with the [ATC add-on tool](#) to monitor special branches or interfaces in addition to ones monitored by setting the [Limit Monitoring Settings](#). This would be done for example if trying to determine an interface flow limit so that interface can be used as a proxy for limiting the transfer between a source and a sink. This same functionality is desirable when using the PV analysis. Both the ATC and PV analyses are similar in that they study the impacts of real power transfers. The ATC analysis is concerned with thermal limitations while the PV analysis looks at voltage stability limitations. Use of the ATC Extra Monitors with the PV analysis could be additionally useful for developing a proxy interface to help prevent voltage stability problems.

Click the **Define ATC Extra Monitors** button to view a list of existing ATC Extra Monitors or create new ones. To include monitoring of these elements at part of the PV analysis, check the **Include ATC Extra Monitors** box.

When monitoring ATC Extra Monitors with the PV analysis, the real power (MW) flow on the specified branches and/or interfaces will be monitored at each base case transfer level. No monitoring is done with contingencies implemented.

ATC Extra Monitors can be added to the [Overview](#) table, available on the [PV Results](#) tab, via the [Display/Column Options...](#) dialog. When displayed in this table, the value reported with the extra monitor is the flow on the extra monitor at the Max Shift level of transfer for the given scenario. Values for ATC Extra Monitors will only be displayed in the Overview table for critical scenarios.

Values for ATC Extra Monitors can also be displayed in the Overview table for transfer levels at which inadequate voltages are found. These values will only be available if using the option found on the [Limit Violations](#) tab to **Store inadequate voltages**. Fields for reporting these values are found in the Inadequate Voltage folder in the list of available fields for the Overview table. The value reported for each extra monitor is the flow on the extra monitor at the **Inadequate Voltage Nominal Shift** transfer level.

Access to the ATC Extra Monitor values stored at transfer levels other than the Max Shift transfer for critical scenarios and values stored for non-critical scenarios is available through SUBDATA sections of the PWPVResultListContainer data type that is saved when saving PV results to an auxiliary file or in the output file saved when choosing to save results to file from the [PV output](#) tab.

Devices at Limits

The **Devices at Limits** sub-tab allows the selection of various options to track devices that hit or back off limits during the PV analysis. Generators and switched shunts at var limits, LTC transformers at tap limits, and lines and interfaces at thermal limits can all be tracked.

Track Hitting and Backing Off Limits during Ramping

<input type="checkbox"/> Generators Var Limits	<input type="text"/>	Define...
<input type="checkbox"/> Switched Shunts Var Limits	<input type="text"/>	Define...
<input type="checkbox"/> LTC Transformers Tap Limits	<input type="text"/>	Define...
<input type="checkbox"/> Lines Thermal Limits	<input type="text"/>	Define...
<input type="checkbox"/> Interfaces Thermal Limits	<input type="text"/>	Define...

Note: Narrow limit tracking by specifying a filter

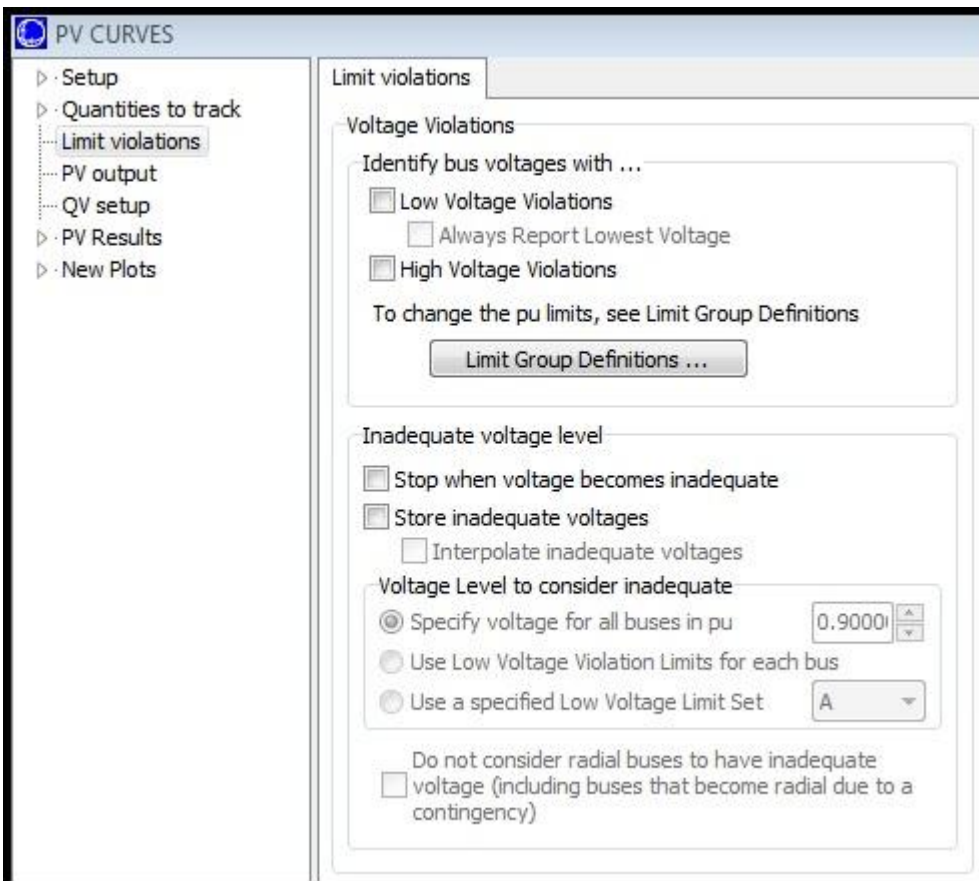
Note: If no filter is specified, then all objects will be tracked.

To track the limits of any of these devices, check the appropriate checkbox. Next to each checkbox there is a dropdown box for selecting an [advanced filter](#) to apply to the tracking. A new filter can be defined for a particular device by clicking the **Define** button next to that device. The filter will limit the number of devices that get tracked. It is a good idea to define a filter for tracking devices so that all elements in the case will not be tracked. Keep in mind that tracking any quantities will take up space in the computer memory. If too many devices are tracked, it is possible to run out of memory.

With these options, device limits are only tracked in the base case at valid transfer levels. No device limit tracking is done with contingencies implemented.

PV Curves: Limit Violations

The Limit violations tab is found on the [PV Curves dialog](#) and allows the definition of what should be considered a violation of a monitored voltage. It also provides specification of parameters that will force the PV process to stop if voltages become inadequate. When choosing to report limit violations in the PV Curve Tool, if no limit violations are found then the minimum voltage is reported instead.



Identify bus voltages with...

These options will store monitored bus voltages that violate either their high or low voltage limit as of the last successful solution for each scenario. Only those buses that are set to be monitored based on the [Limit Monitoring Settings](#) will be monitored, and the low and high voltage limits for each bus are also determined by the limit monitoring settings.

Low Voltage Violations - check this box to store the buses that have low voltage violations

High Voltage Violations - check this box to store the buses that have high voltage violations

Limit Group Definitions - click this button to open the [Limiting Monitoring Settings dialog](#)

When choosing to identify limit violations, the [Overview](#) table on the [PV Results](#) tab will contain information about the number of violations (#Viol), worst voltage violation (Worst V Viol), and bus at which the worst voltage violation occurs (Worst V Bus). Also, for any scenario in which a violation occurs, the **Show Violations** option is enabled on the local right-click menu of the Overview table. Choosing this option will open a table detailing all of the violations.

Inadequate voltage level

Stop when voltage becomes inadequate

Checking this box will stop the PV process for a scenario when any monitored bus voltage falls below the inadequate voltage threshold. The [Limit Monitoring Settings](#) dictate which buses are monitored.

Store inadequate voltages

Checking this box will store all inadequate bus voltages encountered for each scenario. The [Limit Monitoring Settings](#) dictate which buses are monitored. A list of inadequate voltages for each scenario at each transfer level will then be available for viewing as one of the local menu options obtained by right-clicking on the [Overview](#) table (PV Curve records > Show Inadequate Voltages). Inadequate voltages will also be stored in the SUBDATA section of the PWPVResultListContainer DATA section when storing PV results to an auxiliary file.

Interpolate inadequate voltages- If checked add the ability to estimate at what transfer level a voltage becomes inadequate by linearly interpolating between the two transfer levels where it goes from adequate to inadequate.

There are several fields available on the Overview table that provide summary information about the inadequate voltages:

Inadequate Voltage Nominal Shift- nominal transfer level at which the first inadequate voltage is recorded

Inadequate Voltage Worst Voltage- minimum voltage of all buses that have inadequate voltages at the Inadequate Voltage Nominal Shift transfer level

Inadequate Voltage Worst Voltage Bus- bus with the minimum voltage of all buses that have inadequate voltages at the Inadequate Voltage Nominal Shift transfer level

Inadequate Voltage ATC Mon- if any ATC Extra Monitors are being tracked during the analysis, this will list of the flow on the given extra monitor at the Inadequate Voltage Nominal Shift transfer level

If choosing to **Stop when voltage becomes inadequate**, no inadequate voltages will be stored even if this option is selected. When a scenario becomes critical because of inadequate voltages, the results that are given in the Overview table are from the last transfer level at which all voltages were adequate. The inadequate voltage reported in the results is the voltage that would be inadequate if we went beyond the reported transfer level. Because of how the reporting works to report the last non-critical results, it makes sense that there would not be any inadequate voltages stored.

The inadequate voltage threshold when either of the options above is checked is determined by the following options:

Specify voltage for all buses in pu - specify a single inadequate voltage threshold to be used for all monitored buses

Use Low Voltage Violation Limits for each bus - use the limit monitoring settings

Use a specified Low Voltage Limit Set - select a limit set to specify the limit from the drop-down. Up to four (A-D) specific limit sets can be specified for a bus. When using this option, all monitored buses should have the selected limit set specified. If not, the default low voltage limit will be -1.

Do not consider radial buses to have inadequate voltage (including buses that become radial due to a contingency) - check this box to exclude monitoring buses that are radial regardless of whether they are radial because they are connected to the system by a single line or

because they are radial because they are connected to the system via a single in-service line (i.e. they could have more connections but all but one of the connections is open).

This option works somewhat in conjunction with a similar option found on the [Limit Monitoring Settings dialog](#). The option **Do not monitor radial lines and buses** found on the Limit Monitoring Settings dialog will exclude any bus from monitoring if it is radial in the base case. This limit monitoring settings option considers buses to be radial if they are connected by a single branch and only a single branch. When the limit monitoring settings option is selected, radial buses by that definition will be excluded from the PV monitoring. When the PV option is selected, buses that are connected by only a single in-service branch after the application of any scenario contingency, or in the base case if there is no contingency, are considered radial and will be excluded from the monitoring. The PV option is enforced regardless of whether or not the option with the limit monitoring settings is selected.

PV Curves: Output

The Output tab is found on the [PV Curves dialog](#) and allows the designation of where data should be logged.

During the PV analysis, the value of each monitored quantity is recorded at each solved transfer level. This data will only be present in memory unless some output file is specified.

Specifying File for Results

Save results to file

Click this option to save results to a file. In the adjacent text box supply the complete path and name for the output file. If a directory is included with the output file and it does not exist, the directory will be created. If the directory cannot be created, an error will result and the PV analysis will not start. Click the **Browse** button to open a save dialog that will allow selection of the output file. Click the **View** button to open the specified output file in a text editor. The **View** button will be enabled if the specified output file exists.

The results will be saved as a comma-separated variable file regardless of the file extension chosen. Set the file extension to .CSV to make the file type more recognizable by programs such as Excel. If using the save dialog opened with the Browse button, the Save as type field on the save dialog should be set to reflect the desired file extension. Regardless of the file extension chosen, the file is a text file with all contents separated by commas.

When saving results to a file, a log file will also be created that indicates how long the entire PV process took and the different base case and contingency scenarios examined at each nominal shift level. This log file will use the same file name specified for the output file with "_LOG" appended to the end of the file name. The log file will be saved with .TXT extension. If running multiple instances of Simulator for PV analysis, a unique file name must be given to each set of results to avoid overwriting results created from other instances.

Transpose results

Click this option to transpose the columns and rows in each section of the output file. The output file is divided in sections for each of the studied scenarios. The default file format organizes the file so that each row represents a transfer level and each column represents a tracked quantity. The transposed file is formatted such that each row represents a tracked quantity with each column representing a transfer level. This format is useful when the number of tracked quantities exceeds the number of transfer levels.

Single Header File

Additional file formats are available when saving results to file when not transposing the file. If using the default output format (Single Header file option and Transpose results option both unchecked), each scenario will be written as a section with a header line at the beginning of each section indicating what fields are included. Scenarios that become critical during the analysis will be listed with the base case scenario at the shift level at which they became critical. An example of this format is given below:

Example for Default PV Output:

RESULTS FOR PV STUDY "PVSTUDY" *

*Scenario** "base case"

"Nominal", "Export", "Import", "Bus 3 (Three_138.0) PU Volt", "Bus 5 (Five_138.0) PU Volt", "Newly Critical"

0.0000, 0.0000, 0.0000, 0.9927, 1.0066,

100.0000, 100.0000, -98.7365, 0.9928, 1.0062,

200.0000,200.0000,-193.4265, 0.9928, 1.0054,
 300.0000,300.0000, -278.6270, 0.9928, 1.0043,
 400.0000,400.0000, -365.9341, 0.9927, 1.0031,
 500.0000,500.0000,-449.7520, 0.9926, 1.0015,
 600.0000,599.9999,-525.2490, 0.9924, 0.9996,
 650.0000,650.0000,-563.8931, 0.9923, 0.9986,
 675.0000,674.9999,-582.8196, 0.9923, 0.9980,L_00002Two-00006SixC1 (Reached Nose)

If choosing to use a **Single Header File** by checking this option, only a single field header will be shown at the top of the file. The first line of the file indicates that these are the results for a PV study. Following the header are lines indicating scenarios that have become critical during the analysis. Following this information is the single header line that indicates what fields are provided with the results. Following the header are the results for each scenario. An example is given below:

Example Single Header File Output:

RESULTS FOR PV STUDY "PVSTUDY"

"Newly Critical", "L_00002Two-00006SixC1 (Reached Nose)"

"Scenario", "Nominal", "Export", "Import", "Bus 3 (Three_138.0) PU Volt", "Bus 5 (Five_138.0) PU Volt"

base case, 0.0000, 0.0000, 0.0000, 0.9927, 1.0066

base case, 100.0000,100.0000, -98.7366, 0.9928, 1.0062

base case, 200.0000,200.0000,-193.4267, 0.9928, 1.0054

base case, 300.0000,300.0000,-278.6272, 0.9928, 1.0043

base case, 400.0000,400.0000,-365.9344, 0.9927, 1.0031

base case, 500.0000,500.0000,-449.7524, 0.9926, 1.0015

base case, 600.0000,599.9999,-525.2494, 0.9924, 0.9996

base case, 650.0000,650.0000,-563.8935, 0.9923, 0.9986

base case, 675.0000,674.9999,-582.8201, 0.9923, 0.9980

L_00001One-00002TwoC1, 0.0000, 0.0000, 0.0000, 0.9906, 1.0067

L_00001One-00002TwoC1, 100.0000, 100.0000,-98.7366, 0.9882, 1.0062

L_00001One-00002TwoC1, 200.0000, 200.0000,-193.4267, 0.9849, 1.0053

State Archiving

In addition to saving how the monitored quantities vary with the transfer, the entire system state can be saved to file at regular intervals during the analysis. This can be helpful if for analyzing particular transfer levels more closely after the analysis is complete. Keep in mind that, depending on the size of your system, archiving states frequently can require significant disk space and will delay the process.

The state that is saved to file has automatic generation control disabled for the entire case. This is to prevent confusion when these cases are opened. With this option disabled and any changes are made to a case, the only generator that will move will be the system slack. The onus is then on the user to decide if automatic generation control should be enabled for the case and how this will be handled on an area, superarea, or island-wide basis.

When using [Integrated Topology Processing](#), state archiving is available. The state will be archived using the consolidated planning-type model.

The following options are available for archiving states:

Do not save system states

This is the default option. No system states will be saved to file.

Save only the base case for each critical contingency

The base case state without the contingency implemented will be saved to file for each scenario that is critical. The state will be saved at the transfer level that is reported as the critical Max Shift value as given in the [Results](#).

Save all states

Choosing this option will save to file all scenarios at each valid transfer level. Contingency scenarios will be saved with the contingency implemented, unless at the critical transfer level. If a scenario is critical, the base case state without the contingency implemented will be saved at the critical transfer level. For critical scenarios, the critical transfer level is the value that is reported as the Max Shift value given in the [Results](#). The base case state with just the transfer implemented will also be saved for all scenarios at all transfer levels at which the scenario will solve.

Save state as

Use this drop-down to select the file type that should be used when archiving states. Files can be saved as [PowerWorld auxiliary files \(*.aux\)](#), [PowerWorld binary files \(*.pwb\)](#), or both.

Where should they be stored?

Specify the directory where state files should be saved. Click the **Browse** button to open a dialog to use for selecting the directory.

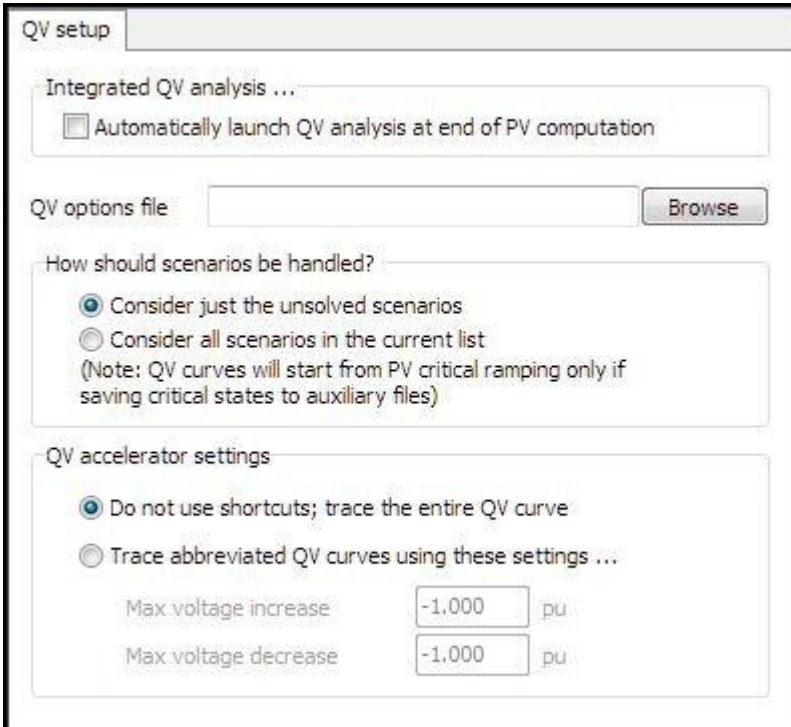
Specify a prefix to use in naming the state archives

Saved state files will be distinguished by a naming convention based on the scenario name and transfer level or critical state indicator and scenario name. Use a prefix to further distinguish these files if saving multiple study results to the same directory.

PV Curves: QV Setup

The QV Setup tab is found on the [PV Curves dialog](#). This page provides options for setting up a [QV analysis](#) and for automatically launching the QV analysis upon completion of the PV analysis.

When using [Integrated Topology Processing](#), the automatic launching of the QV analysis is disabled.



The screenshot shows the 'QV setup' dialog box. It has a title bar 'QV setup'. Inside, there's a section 'Integrated QV analysis ...' with a checkbox 'Automatically launch QV analysis at end of PV computation'. Below that is a 'QV options file' field with a 'Browse' button. Then, a section 'How should scenarios be handled?' with two radio buttons: 'Consider just the unsolved scenarios' (selected) and 'Consider all scenarios in the current list'. A note below says '(Note: QV curves will start from PV critical ramping only if saving critical states to auxiliary files)'. Finally, a 'QV accelerator settings' section with two radio buttons: 'Do not use shortcuts; trace the entire QV curve' (selected) and 'Trace abbreviated QV curves using these settings ...'. Below the second radio button are two input fields: 'Max voltage increase' with value '-1.000' and unit 'pu', and 'Max voltage decrease' with value '-1.000' and unit 'pu'.

The PV curve tool stresses the system by increasing a transfer between injection groups. Eventually, the transfer is increased so much that the power flow can no longer be solved. Under the assumptions of static voltage stability assessment, the point at which the system becomes unsolvable is regarded as the point of voltage collapse. Voltage collapse tends to be a localized phenomenon associated with a lack of reactive voltage support at a bus or group of connected buses. It is important to identify this group of stressed buses so that efforts can be properly directed to reinforce the system. QV analysis is often used to identify the buses that are most prone to a voltage collapse because they are close to having insufficient reactive support.

The design of the PVQV add-on to Simulator acknowledges the important role that QV analysis plays in identifying the buses that are most heavily stressed by a transfer. The approach is straightforward. First, perform a PV curve analysis that models a transfer from source to sink for both base and contingency topologies. This will yield a number of PV curves that terminate at a transfer level that causes the system to become unsolvable for each topology. For each of these critical transfer level / contingency pairs, perform a QV analysis at a set of buses to try to determine which area or areas constitute the epicenter of the collapse. The set of buses at which the QV analysis is performed can be predefined by the user and supplemented by the list of buses that have the lowest voltage magnitudes or the highest VQ sensitivities.

To specify that the QV analysis automatically be launched after finishing a PV study, check the box labeled **Automatically launch QV analysis at end of PV computation**. After the PV curve has found all the critical cases it has been asked to identify, it will launch the [QV Curve Tool](#). The buses

the QV Curve Tool will analyze will then come from two groups: those that the user has pre-selected, and those that the tool automatically identified because they are among the lowest-voltage buses or have the highest VQ sensitivities. Whether or not to include the lowest-voltage buses or highest VQ sensitivity buses are options that need to be set by the user. Pre-selected buses for the QV analysis must also be set by the user. All of these options need to be specified in the **QV options file** that is required to be specified in order to automatically launch the QV analysis.

The remainder of the options that must be specified in order to perform the QV analysis automatically are described below:

QV options file

Specify the name of the file that contains settings for a variety of options associated with the QV study in this text box. Use the **Browse** button to locate this file using a file dialog. The QV options file should specify which buses should be selected for which to draw QV curves.

Among the topics that may be addressed by the QV options file is whether buses should be automatically selected as buses for which to draw QV curves based on their voltage or VQ sensitivity. The easiest way to build a QV options file is to use the [QV Curve Tool](#) to specify the settings of the various options and to save these settings in a file.

How should scenarios be handled?

QV curves can be drawn for the selected buses either for all the scenarios that the PV tool has processed (select **Consider all scenarios in the current list**) or just for the scenarios that the PV Curve tool determined to be critical (select **Consider just the unsolved scenarios**).

The transfer level at which a scenario is studied with the QV curve tool is based on other options that can be set with the PV tool. If choosing to archive states, and at a minimum archiving the critical scenarios, as set on the [Output](#) tab, critical scenarios will be studied at the last transfer level at which each scenario was successfully solved. Another condition is that the states must be saved as PowerWorld auxiliary (*.aux) files.

If not choosing to archive critical states, the critical state auxiliary file does not exist, or studying non-critical scenarios, the transfer level at which a scenario is studied in the QV curve tool is the last transfer level that the PV tool attempted to model. This transfer level is the level that is in place in the state that the case is left in upon completion of the PV analysis.

QV accelerator settings

This group of controls allows the definition of a voltage window over which each QV curve should be calculated.

Do not use shortcuts; trace the entire QV curve

Each QV curve will be computed over the voltage range specified for the bus in the **QV options file**. This voltage range may be specific to that bus or may be a window what has been defined for the entire run.

Trace abbreviated QV curves using these settings...

The voltage range over which each bus's QV curve will be computed will be customized according to the values specified for **Max voltage increase** and **Max voltage decrease**. For example, if a bus is currently at 0.9 pu voltage for a particular scenario and the max voltage increase is specified to be 0.1 with the max voltage decrease specified to be 0.3, the QV curve for that bus for that scenario will be traced over the voltage range 0.6 pu to 1.0 pu.

PV Curves: PV Results

The PV Results page is found on the [PV Curves dialog](#). This is the page from which the PV analysis is launched and provides access to all of the results. This page is divided into some general information at the top and sub-tabs at the bottom that provide details about the results.

PV Results

Run Stop

There are no more limiting cases left to find.

Present nominal shift 0.000

Present step size

Found 1 limiting case.

	Gen MW	Load SMW	Load IMW	Load ZMW
Source	180.05	0.00	0.00	0.00
Sink	0.00	0.00	0.00	0.00

View detailed results

Other actions >>

Run

Click this button to start the analysis.

Stop

Click this button to stop the analysis once it has been started. This button will only be enabled if the analysis has been started.

Status information

Status messages will be displayed as scenarios are examined and the power transfer changes.

Summary information about the transfer

The **Present nominal shift** and **Present step size** fields will provide information about the present power transfer being examined. **Gen MW** and **Load MW** totals for both the source and sink injection group will also update as the power transfer changes to give an indication of the status of each group.

View detailed results

Click this button to open a text editor that contains detailed solution results. The results are in the same format as the output file described on the [Output](#) tab.

Other actions >>

Clicking this button opens a menu that provides access to a number of other options:

View activity log

This option will be enabled if choosing to save the output to a file via options found on the [Output](#) tab. If enabled and selected, the log file will be displayed.

View detailed results

This option does the same thing as the **View detailed results** button described above.

Clear results

This option will destroy all of the results. Nothing is done to the system state.

Save critical contingencies

Only those contingencies for critical scenarios will be saved to an auxiliary file. Selecting this option will open a dialog prompting for a file name. In addition to contingency definitions,

additional data including limit monitoring settings, contingency solution options, and power flow solution options will be saved.

Restore initial state

When the PV analysis first starts running, the present system state is stored in memory and identified as the initial state. Choose this option to return back to the initial state.

When the analysis completes, the state that the case is left in is the last analyzed scenario with the transfer in place but any contingency removed (i.e. base case state at the critical transfer level). If the last analyzed scenario is a reverse transfer scenario, the system state that is in place is the base case state in place when the PV analysis first starts running. The reverse transfer process always restores to the base case state upon completion.

The initial state can also be set by the **Set current state as initial** option. If this is done, choosing to restore initial state will return back to whatever the set initial state was.

Restore last solved state

When the analysis completes, the state that the case is left in is the last analyzed scenario with the transfer in place but any contingency removed (i.e. base case state at the critical transfer level). This option can be used to set the state to the last solved state. The last solved state is the base case state at the last transfer level at which it was studied and solved. This transfer level will most likely be higher than the transfer level at the last studied critical scenario. If the last analyzed scenario is a reverse transfer scenario, the last solved state is the base case state in place when the PV analysis first starts running.

The way that the process works when ramping in the positive direction is to ramp the transfer by the specified step size in the base case. The base case state at the last solved ramped amount becomes the last solved state. Then all contingency scenarios are tested. If any contingency does not solve at the present transfer level, the contingency is studied individually to determine the transfer level at which it becomes critical. This transfer level will most likely be less than the last solved state transfer level.

Set current state as initial

Choose this option to set the initial state to be the current state. This might be useful if doing analysis outside of the PV tool and needing to be able to return to the present state. The **Restore initial state** option can then be used to return to the state designated as the initial state. This option might also be useful if desiring to do another analysis starting from the present state and choosing to do this by using the **Start over** option. The Start over option will restore back to the initial state before starting any analysis so it is important that the initial state be set correctly.

Keep in mind that once the **Run** button is clicked to start the analysis, whatever state is presently in place will be designated as the initial state.

Start over

This option will destroy any existing results, clear the log, clear all plots and plot controls, and restore the system state to the initial state. See comments in the **Restore initial state** and **Set current state as initial** options for more details on the initial state.

Run QV tool

This option will launch the QV analysis by opening the [QV Curves dialog](#) and then starting the analysis. The options specified on the [QV setup](#) tab will be used for the analysis.

This option will not be enabled if using [Integrated Topology Processing](#).

Details about the sub-tabs at the bottom of page can be found at the following:

[Overview](#)

[Plot](#)

[Track Limits](#)

PV Curves Results: Overview

The Overview sub-tab is found on the [PV Results](#) tab of the [PV Curves dialog](#). This page provides a summary of all scenarios studied and their results. This table will update as the analysis progresses.

Overview Plot Track Limits						
	PV Scenario	Critical?	Critical Reason	Max Shift	Max Export	Max Import
1	base case	NO				
2	L_000001One-000002TwoC1	YES	Reached Nose	1162.50	1162.50	-904.15
3	L_000001One-000003ThreeC1	NO				
4	L_000002Two-000003ThreeC1	NO				
5	L_000002Two-000004FourC1	NO				
6	L_000002Two-000005FiveC1	NO				
7	L_000002Two-000006SixC1	YES	Reached Nose	675.00	675.00	-584.41
8	L_000003Three-000004FourC1	NO				
9	L_000004Four-000005FiveC1	NO				
10	L_000007Seven-000005FiveC1	YES	Reached Nose	2550.00	2550.00	-1463.22
11	L_000006Six-000007SevenC1	YES	Reached Nose	2537.50	2537.50	-1440.64
12	L_000006Six-000007SevenC2	YES	Reached Nose	2550.00	2550.00	-1439.07

The overview is a [case information display](#) and therefore can be used in a manner consistent with all other case information displays.

There are a number of fields that are shown by default on the display:

PV Scenario

Name of the scenario. Either *base case* or the contingency name.

Critical?

YES if the scenario is considered to be critical. Otherwise, *NO*.

Critical Reason

This will be blank as long as the scenario is not critical. If the scenario is critical, there are a number of reasons that could be displayed explaining why the scenario is critical:

Reached Nose - The power flow failed to solve so the system is considered to be at the nose of the PV curve.

Inadequate Voltage - The process stopped because at least one voltage is considered inadequate. This reason will only occur when using the option specified on the [Limit Violations](#) tab to **Stop when voltages become inadequate**. The message will be appended with the bus and voltage of the bus with the lowest inadequate voltage.

Source Maxed - The source no longer has enough resources to continue the power transfer. Source resources are affected by the options **Enforce unit MW limits** and **Do not allow negative loads** found on the [Common Options](#) sub-tab of the Setup tab. They could also be affected if choosing to use merit order dispatch in which unit limits are always enforced.

Sink Maxed - The sink no longer has enough resources to continue the power transfer. Sink resources are affected by the options **Enforce unit MW limits** and **Do not allow negative loads**

found on the [Common Options](#) sub-tab of the Setup tab. They could also be affected if choosing to use merit order dispatch in which unit limits are always enforced.

SRT - Indicates that a solvable reverse transfer has been found when choosing to use the reverse transfer option found on the [Advanced Options](#) sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

RT Source Maxed - Indicates that the source no longer has enough resources to continue the power transfer when using the reverse transfer option found on the [Advanced Options](#) sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

RT Sink Maxed - Indicates that the sink no longer has enough resources to continue the power transfer when using the reverse transfer option found on the Advanced Options sub-tab of the Setup tab. This message is appended with the original critical reason that required the use of the reverse transfer.

Reached Nose in Base Case - This message will only occur for a contingency scenario. It indicates that the contingency scenario is actually less limiting than the base case scenario and that the analysis of the contingency scenario cannot continue because of the base case limitation. The power transfer is always ramped in the base case prior to implementing the contingency. The contingency cannot be attempted unless the base case solves with the transfer implemented.

Max Shift

This value will only be filled in for critical scenarios. This is the nominal shift that was implemented to reach the critical scenario. The nominal shift is the actual accumulated transfer that was attempted.

Note: With a few exceptions, shift values (includes Max Shift, Max Export, and Max Import) that are reported are from the last solved solution before the critical point was reached. Because the critical point is not considered a valid solution, no valid shift can be determined at this point. Therefore, the values that are reported are as close to this point as the process can get within the tolerances specified for the transfer step size. Exceptions to this occur when using the reverse transfer option. In this case, the values that are reported are for the minimum transfer level at which the contingency will solve and all voltages are adequate. Another exception occurs when a scenario becomes critical because of inadequate voltages at the zero transfer level. Because the scenario can be solved at this point, the shift values will be recorded at the actual step at which the voltages are inadequate. If a case becomes critical due to inadequate voltages and is not at the zero transfer level, the shift values reported for these will follow the rule that they come from the last solved step at which all voltages were adequate. The recording of values determined by [Quantities to Track](#) will also follow these same rules.

If a scenario is critical because either the source or sink does not have enough resources to continue the power transfer, the shift values and tracked quantities that are reported are at the maximum transfer level that can be achieved without source and sink limits being violated.

Max Export

This value will only be filled in for critical scenarios. This is the amount that the injection changed in the source injection group due to the transfer and contingency implementation. This value's magnitude can differ from the Max Shift value due to injection changes resulting from make-up power requirements from the contingency.

See note with Max Shift for additional explanation on how shift values are recorded.

Max Import

This value will only be filled in for critical scenarios. This is the amount that the injection changes in the sink injection group due to the transfer, loss changes due to the transfer, and contingency implementation. This value's magnitude can differ from the Max Shift value due to the injection

changes resulting from make-up power requirements from the contingency and making up for losses due to the transfer. The sink injection group is designated to account for all losses due to the transfer.

See note with Max Shift for additional explanation on how shift values are recorded.

#Viol, Worst V Viol, Worst V Bus

These fields will be filled when choosing to monitor low or high voltage violations and there are violations for a given scenario. The options to monitor voltage violations are found on the [Limit violations](#) tab. These fields give the total number of all bus violations for a scenario, the voltage of the worst violation, and the bus at which the worst violation occurs, respectively. The worst violation is considered to be the one with the lowest voltage.

Max P Mism Bus #, Max P Mism Bus Name, Max MW Mism

These fields will only be filled in for critical scenarios. These give the bus number, bus name, and real power mismatch for the bus with the maximum real power mismatch for the power flow solution at which the scenario becomes critical. For scenarios that are at the nose of the PV curve, the mismatch should reflect the fact that the power flow solution will not converge and might give some insight into problem areas of the system.

Max Q Mism Bus #, Max Q Mism Bus Name, Max Mvar Mism

These fields will only be filled in for critical scenarios. These give the bus number, bus name, and reactive power mismatch for the bus with the maximum reactive power mismatch for the power flow solution at which the scenario becomes critical. For scenarios that are at the nose of the PV curve, the mismatch should reflect the fact that the power flow solution will not converge and might give some insight into problem areas of the system.

The right-click local menu has several options specific to PV results:

Show Dialog

Choosing this option will open the [Contingency Definition dialog](#) for the associated scenario's contingency. This dialog can be used to examine the contingency and make any necessary changes.

PV Curve records

There are three PV tool specific options available under this menu item:

Show Violations...

This option is available when choosing to monitor low or high voltage violations and there are violations for a given scenario. The options to monitor voltage violations are found on the [Limit violations](#) tab. When monitoring voltage violations, any bus voltages that violate limits will be stored for each scenario at the last studied transfer level. Choose Show Violations... for a given scenario to display a table of buses that have voltage violations.

Restore critical state

This option will restore the system state to the selected critical state. This means that the critical transfer and any associated contingency will be implemented. This option is only available if choosing to archive system states as set on the [PV output](#) tab. At a minimum the option to save critical states must be selected. Another condition is that the states must be saved as PowerWorld auxiliary (*.aux) files.

Because state archiving is disabled when using [Integrated Topology Processing](#), this option to restore critical states is also disabled.

Show Inadequate Voltages...

This option is available when choosing to store inadequate voltages and there are inadequate voltages for a given scenario. The option to store inadequate voltages is found on the [Limit violations](#) tab. Choose Show Inadequate Voltages... to open a table listing any bus that has an inadequate voltage for any transfer level for the given scenario and the voltage for each listed bus at each studied transfer level. If a bus voltage is blank, that means the voltage at that bus was not inadequate for that transfer level.

Calculated Fields

[Calculated fields](#) are available for use with the PV results. The object type that is used for the PV results is **PV Curve**. This type of calculated field can only be used with PV results. Calculated fields are useful when doing advanced filtering that might involve returning, for example, only the scenario with the lowest inadequate voltage at the lowest inadequate transfer level. Calculated fields are needed to return the lowest inadequate transfer level and then the lowest inadequate voltage that occurs at a transfer level less than or equal to the lowest inadequate transfer level. This type of filtering is needed because it would be possible for inadequate voltages to occur for different scenarios at the same transfer level. The most critical would then be interpreted as the scenario that produces the lowest inadequate voltage.

PV/QV Curves Results: Plot

Plotting functionality is almost identical for both the PV analysis and QV analysis (exceptions will be noted below). For PV analysis, the Plot tab is accessed from the [PV Results](#) tab of the [PV Curves dialog](#). For QV analysis, the Plots tab is accessed from the [QV Results](#) tab of the [QV Curves dialog](#). Any of the values selected to be [monitored](#) can be plotted using this display.

Horizontal axis value ☐ Pre-contingency
Nominal Shift

Vertical axis value type
PU Volt

Plot values for these elements ...

Description	Plot?
Three (3)	NO
Five (5)	NO

For these scenarios ... ☐ Always include base case

PV Scenario	Critical?	Plot?
base case	NO	NO
L_000001One-000002TwoC1	YES	NO
L_000001One-000003ThreeC1	NO	NO
L_000002Two-000003ThreeC1	NO	NO
L_000002Two-000004FourC1	NO	NO
L_000002Two-000005FiveC1	NO	NO
L_000002Two-000006SixC1	YES	NO
L_000003Three-000004FourC1	NO	NO
L_000004Four-000005FiveC1	NO	NO
L_000007Seven-000005FiveC1	YES	NO
L_000006Six-000007SevenC1	YES	NO

Plot title

Clear Plot

The page is broken into two basic sections: the left and right half. The right half of the display allows selection of the scenarios that should be included on a given plot. The left half allows selection of the quantities that will be charted on the plot.

Selecting Scenarios to Plot

The right half of the display gives a table of all scenarios that were studied. Change the **Plot?** field to **YES** to include a scenario in the current plot. The Plot? field is a toggleable field and can be changed from **YES** to **NO** and back again by double-clicking on the appropriate entry. Any scenario whose Plot? field is set to **YES** will be included on the same plot once the **Plot** button is clicked.

The **Always include base case** box should be checked if the base case scenario should be included in the plot along with any other scenarios selected. This option is not available for QV results plotting.

Before a plot can be created, the quantities to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Selecting Quantities to Plot

The left half of the display gives the various options for selecting what quantities should be plotted.

The **Horizontal axis value** drop-down contains any parameters that were monitored. Additionally, for the PV analysis, this drop-down also contains the independent quantities (Nominal Shift, Export, and Import) recorded during the analysis. If the **Pre-contingency** box is checked, the x values for each plotted scenario will come from the base case instead of the results of a given scenario. The Pre-contingency check box is not available for QV results plotting.

The **Vertical axis value type** drop-down contains a list of all the types of values that were monitored. The type selected in this drop-down will determine the actual elements that are listed in the **Plot values for these elements** table. To include an element in the plot, change the **Plot?** field to **YES** for that element. The Plot? field is a toggleable field and can be changed from **YES** to

NO and back again by double-clicking on the appropriate entry. Any element whose Plot? field is set to *YES* will be included in the same plot once the **Plot** button is clicked.

Before a plot can be created, the scenarios to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Plot Title

Optional field that can be set to add a title to a plot and distinguish one plot from another.

Plot

After selecting the scenarios and values to plot, click this button to actually generate the plot. The plot will appear in its own window. The number of plots that can be created is limited only by computer system memory.

Right-clicking on a plot will expose a local menu with several options. The plot can be saved as a bitmap, Windows metafile, JPEG, or text file by clicking **Save** and selecting the appropriate file type. **Copy** will allow the plot to be copied and pasted into other applications. **Print** will send the plot to a printer that can be configured by selecting the **Printer Setup** option. Selecting **Close** will close the plot window.

Clear

Click this button to reset all of the **Plot?** fields to *NO* for both the scenarios to plot and the values to plot.

PV Curves New Plots

The New Plots page is found on the [PV Curve dialog](#). The Plots page is broken into two sub-tabs that provide the means for creating and modifying plots: [Plots Page](#)

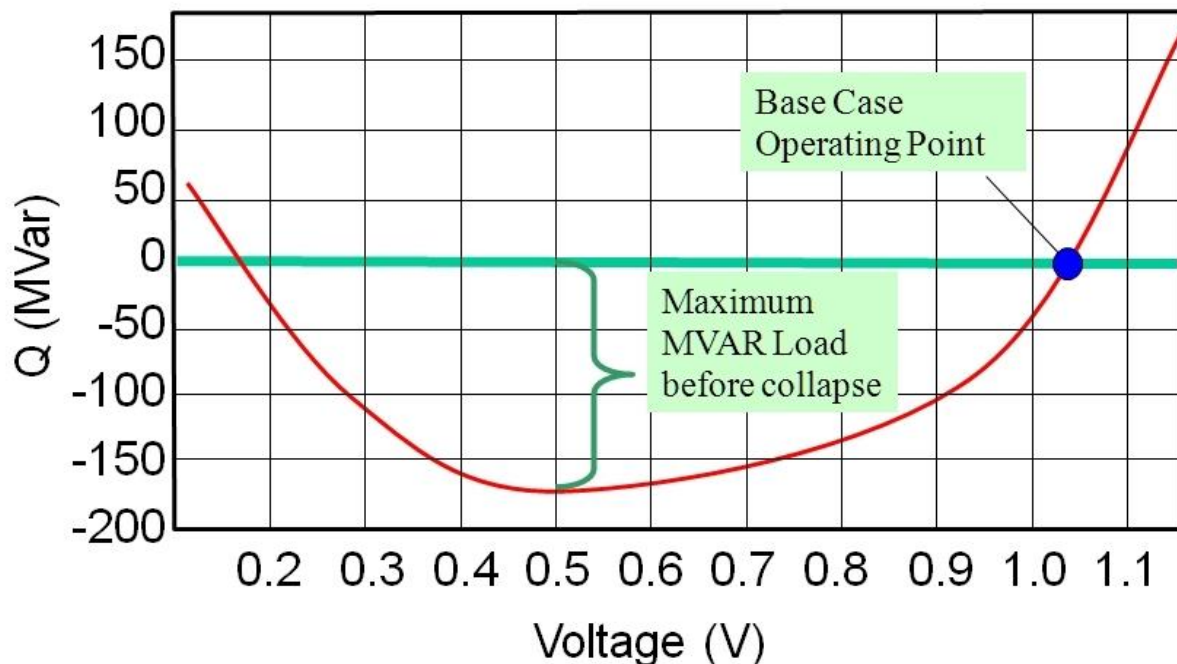
QV Curves

The QV Curves tool provides the ability to compute QV curves for any bus in the system. A QV analysis studies how variations in reactive power (Q) injection at a bus affects the voltage (V) at that same bus. Other system parameters can also be monitored as the reactive power injection changes.

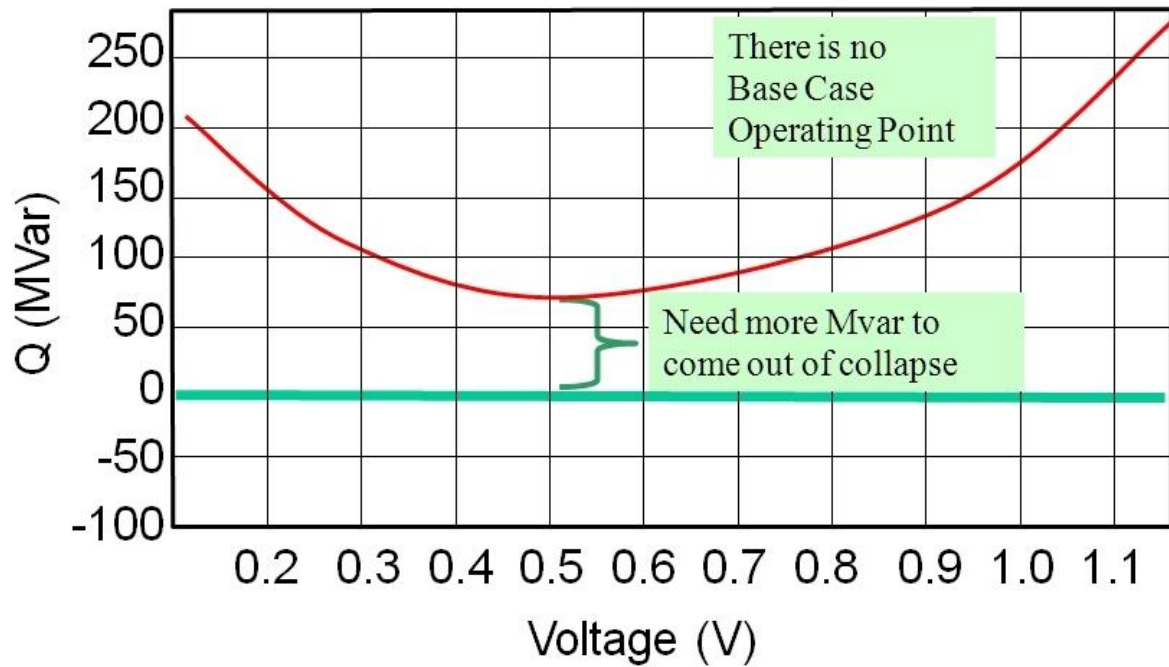
To create a QV curve, a fictitious generator (synchronous condenser) is placed at a bus that is being studied. The voltage set-point of this generator is varied and its var output is allowed to be any value needed to meet this voltage set-point. The vertical axis (y-axis) of a QV curve depicts the output of the fictitious generator in Mvar. The horizontal (x-axis) depicts the respective voltage under this condition. The base case operating point of the system is represented by the x-intercept of the curve. This is the point where the fictitious generator is at 0 Mvar output and represents the base case. There are situations in which the output of the fictitious generator is not 0 Mvar in the base case, but these will be explained with the appropriate option settings. **When considering a contingency scenario in the QV analysis, the term *base case operating point* will also be used. This is the starting case that represents the system prior to performing any analysis for the curve tracing.**

Tracing down the curve from higher to lower voltage set-points represents a decrease in the fictitious generator's Mvar output which is representative of an increase in Mvar load. The curve is then tracing what the voltage would be as the Mvar load increases. At some point the Mvar value of the generator will stop decreasing and the bottom of the curve will be reached. This point represents the maximum increase in load Mvar that can occur at this bus before voltage collapse is reached.

The following shows a typical QV curve. The plot is actually a VQ curve, but this is traditionally called the QV curve.



The following plot shows a situation in which there is not enough Mvar reserve. There is no base case operating point as the curve never crosses the x-axis. The difference between the bottom of the curve and the x-axis is the amount of Mvar injection needed to achieve a solvable case and to come out of collapse.



To use the QV Curves tool, select **QV Curves** on the **PV and QV Curves (PVQV)** ribbon group on the [Add Ons](#) ribbon tab. The [QV Curves dialog](#) will open from which buses to be monitored can be specified, options for the QV curves can be set, and the QV curve analysis can be run.

QV Curves Dialog

To display this dialog, go to the [Add Ons](#) ribbon tab and select **QV Curves** from the **PV and QV Curves (PVQV)** ribbon group.

The QV Curves dialog contains all of the setup and controls for processing and analyzing the QV curve analysis. The dialog is broken down into several pages:

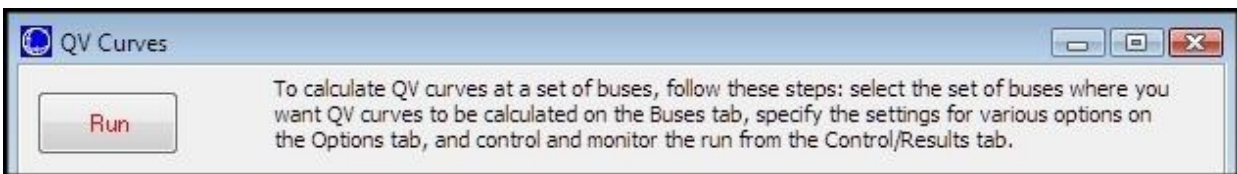
[Buses](#)

[Quantities to Track](#)

[Options](#)

[Results](#)

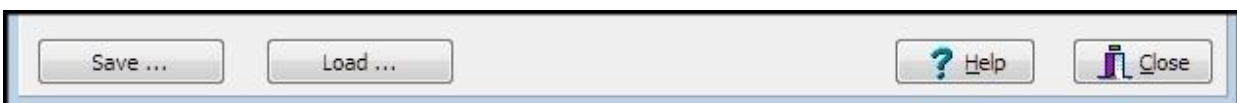
There is a section at the top of the dialog containing the **Run** button that will be displayed regardless of the page that is selected.



When all options have been set, click **Run** to start the QV analysis. The progress of the analysis can be monitored from the **Results** tab. While the analysis is in progress, the Run button will become a **Stop** button that can be clicked to stop the analysis.

When running the QV analysis, the first step is to make sure that the base case is solvable. If it is not solvable, a message will prompt the user to decide if a solvability analysis be done of the base case. A solvability analysis on the base case will process each of the buses selected for analysis. As each bus is processed, the fake generator (synchronous condenser) that is normally added to trace the QV curve at a bus will first be used to determine if Mvar injection/absorption can be adjusted such that a solvable point is found. This process can be quite time consuming. The process first tries to find a solvable point by increasing the studied setpoint voltage and tracing up the curve up to the maximum voltage. It is more likely that a solvable point will be found tracing up the curve due to the increased Mvar injection required to meet the setpoint voltage. If a solvable point is found, this solvable point with the fake generator injection/absorption in place will then be the starting point for the QV curve tracing. Because the base case is not solvable, no attempt will be made at solving any contingency scenarios. If the base case is not solvable and the solvability analysis is not done, the QV process will stop.

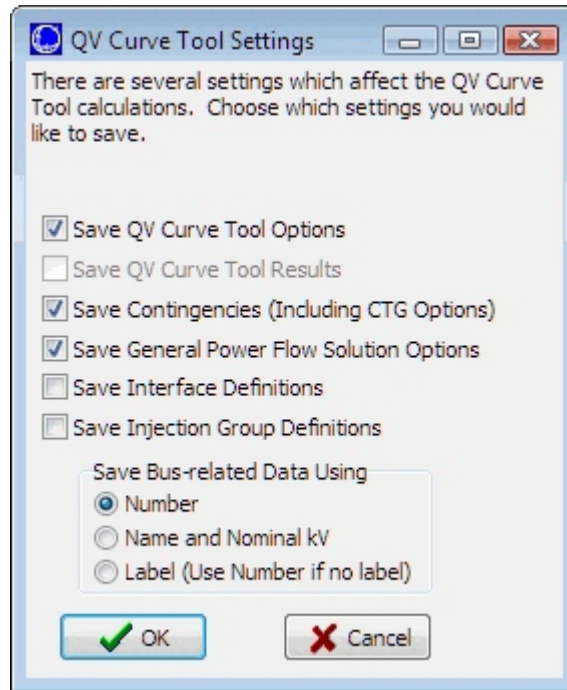
There are also several buttons at the bottom of the dialog that are available regardless of the selected page.



Save ...

Clicking this button will prompt for a filename in which to save an [auxiliary file](#). QV-specific results and option settings are not saved with the case when it is saved to file. Use this option to store any results and settings that need to be retained for future use.

The QV Curve Tool Settings dialog shown below will be displayed with options for specific data to include in the auxiliary file. Check the box next to a particular data set to save this data in the auxiliary file. Which key field to use when identifying objects in the file can also be specified on this dialog. Click **OK** on this dialog to finalize saving the file or **Cancel** to abandon the file save.



Load ...

Clicking this button will open a dialog to allow the user to select an [auxiliary file](#) to load. This is intended to be used for loading any relevant option settings to be used during the QV analysis. The dialog will be updated according to the option settings loaded from the file.

Help

Displays this help page.

Close

This will close the dialog without running the analysis. The currently set options will be saved, and the dialog will not close unless all option settings are valid. Any results will remain in memory as long as the power flow case remains in memory. QV curve results and option settings are not saved with the case when it is saved to file.

QV Curves: Buses

The Buses page is found on the [QV Curves dialog](#).

Buses

	Number	Name	Nom kV	Selected?	Min Volt	Max Volt	Step Size
1	1	One	138.00	NO			
2	2	Two	138.00	NO			
3	3	Three	138.00	NO			
4	4	Four	138.00	NO			
5	5	Five	138.00	NO			
6	6	Six	138.00	NO			
7	7	Seven	138.00	NO			

Note: The active defined contingencies affect the selection of super buses and resulting selected buses.

To add buses to the QV analysis, specify ...

A range of bus numbers

or the name of a bus

☐ Additionally, automatically draw curves for

0 lowest-voltage buses and

0 highest dv/dq buses

from limit group

The Buses page is used to designate the buses at which QV curves should be calculated. Depending on other options that are selected, a QV curve will be determined for each selected bus for the base case and any selected contingencies. Each bus/contingency (or base case) pair comprise a QV scenario.

By default, the table lists all buses in the case. This table is a bus case information display and [filtering](#) can be applied to this table to make the selection of buses easier. Several QV-specific fields are listed by default:

Selected?

Set this field to *YES* to calculate a QV curve for this bus. This field can be toggled between *YES* and *NO* by double-clicking on the appropriate entry.

Min Volt

Specify the minimum voltage at which the analysis will be performed for this particular bus. If this is not specified, the default minimum voltage set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Max Volt

Specify the maximum voltage at which the analysis will be performed for this particular bus. If this is not specified, the default maximum voltage set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Step Size

Increment between voltage set-points used to trace the curve for this particular bus. If this is not specified, the default step size set with the default [options](#) is used. Only non-default values will be shown. If the value is changed to the default setting, the cell will be set blank.

Modify Selected Buses to Select Only Single Bus Per Super Bus

This button will only be visible if the [Integrated Topology Processing](#) add-on is available. When using topology processing, only a single bus per super bus will be in the consolidated case. That means that the parameters for each bus in the super bus will be the same, i.e. same voltage magnitude and angle, etc. This button can be used to eliminate any excess QV curves that would just take up computer memory by drawing essentially the same curve multiple times for the same super bus. To take advantage of this option and only select a single bus per super bus, first set up the bus selection to monitor all of the necessary buses and then push the **Modify...** button. The Selected field in the table will then be updated to only include a single bus per super bus. When determining the primary node (the primary node is the bus that will ultimately be the one monitored during the analysis) and super buses, the active defined contingencies will be used to determine how the case is consolidated. Because the consolidation can be different based on the contingency set, or lack thereof, it is best to just define the selected buses to include all necessary buses and use this option to refine the list down to a single bus per super bus rather than manually trying to maintain such a list.

Buses can also be selected for analysis by using several options found at the bottom of the dialog. A **range of bus numbers or the name of a bus** can be specified in the appropriate boxes. Clicking the **Add** button for either entry will set the **Selected?** field to *YES* for the designated buses. These options are used to make setting the Selected? field more convenient.

There is another set of options that will have the QV tool determine which buses should be analyzed in addition to those already selected.

Additionally, automatically draw curves for

Check this box to optionally draw additional curves based on certain conditions. Set the number of **lowest-voltage buses** and/or **highest dv/dq buses** at which curves should be traced. The dv/dq sensitivity is the sensitivity of voltage at a bus to an injection of reactive power at the same bus. The sensitivities are ranked based on their magnitude. These lowest-voltage buses and highest dv/dq buses are determined for each scenario once any contingency has been applied. Use the **from limit group** drop-down to specify which buses should be considered when evaluating these conditions. Only buses that are assigned to the selected [limit group](#) will be considered.

If the buses identified when evaluating these conditions already have been selected by the user, no additional buses will be selected to make sure that the number of additional buses to include is achieved by buses that have not been user selected.

QV Curves: Options

The Options tab is found on the [QV Curves dialog](#). The Options page contains several additional sub-tabs from which parameters necessary for carrying out a QV analysis can be set:

[Solution](#)

[Contingencies](#)

[Output](#)

The **Restore Defaults** button is available on the Options tab regardless of the sub-tab that is selected. Clicking this button will set all options found on any of the sub-tabs to the default settings.

QV Curves Options: Solution

These options are found on the Solution sub-tab located on the [Options](#) page of the [QV Curves dialog](#).

Solution Contingencies Output

Default solution parameters

These are the default parameters to use for computing the QV curves. Different values for these parameters can be assigned for each bus.

Voltage step size (pu) 0.0100

Minimum voltage (pu) 0.1000

Maximum voltage (pu) 1.1000

☐ Use initial voltage as Vmax (applies to all buses)

Power flow solution options

Global solution options ...

QV Power Flow Solution Options...

Default solution parameters

Voltage step size (pu)

Increment between voltage set-points used to trace a QV curve. This is the default value that will be applied to a bus unless a bus-specific value is specified.

Minimum voltage (pu)

Minimum voltage at which the analysis will be performed. This is the default value that will be applied to a bus unless a bus-specific value is specified.

There are three ways to set the minimum voltage that will be used for a particular bus. The default value will be used unless a bus-specific value is specified. A third option is possible if launching the QV analysis upon finishing the PV analysis and choosing to use QV accelerator settings. If using the accelerator settings, the bus-specific minimum voltage will be set based on the accelerator settings. More information is available about launching the QV analysis from the PV analysis and setting the accelerator settings on the [QV Setup](#) tab of the [PV Curves dialog](#).

Maximum voltage (pu)

Maximum voltage at which the analysis will be performed. This is the default value that will be applied to a bus unless a bus-specific value is specified.

There are three ways to set the maximum voltage that will be used for a particular bus. The default value will be used unless a bus-specific value is specified. A third option is possible if launching the QV analysis upon finishing the PV analysis and choosing to use QV accelerator settings. If using the accelerator settings, the bus-specific maximum voltage will be set based on the accelerator settings. More information is available about launching the QV analysis from the PV analysis and setting the accelerator settings on the [QV Setup](#) tab of the [PV Curves dialog](#).

Use initial voltage as Vmax

Check this box to use the initial voltage at a bus as the maximum voltage at which the analysis will be performed. This option overrides any of the other options that set the maximum voltage for a bus. The initial voltage at a bus is determined prior to starting the QV analysis but after applying contingencies and applying any additional var support necessary to make an unsolvable scenario solvable.

Power flow solution options

Global solution options

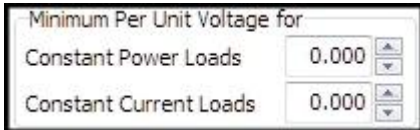
Click this button to bring up the [Simulator Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affect all power flow computations, even those outside the QV process.

The global solution options will be used in the QV analysis unless QV-specific solution options are defined using the option described below. If contingency-specific solution options are defined for a contingency, those options will override both the QV-specific solution options and the global solution options.

QV Power Flow Solution Options

Click this button to bring up a dialog that allows specification of QV-specific power flow solution options. These are the same options that are described in the Simulator Options Dialog for [Power Flow Solution Options](#). The QV-specific solution options will override the global solution options. If contingency-specific solution options are defined for a contingency, those options will override both the QV-specific solution options and the global solution options.

Because the goal is to stress the system while performing the QV analysis, there are two power flow solution options that are set internally as part of the QV process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.



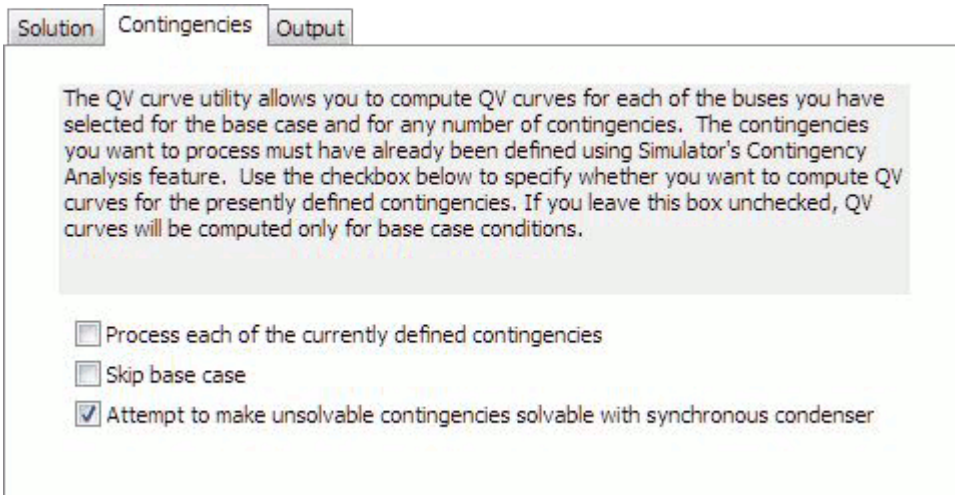
A screenshot of a software dialog box titled "Minimum Per Unit Voltage for". It contains two rows of controls. The first row is labeled "Constant Power Loads" and has a text input field containing "0.000" followed by up and down arrow buttons. The second row is labeled "Constant Current Loads" and also has a text input field containing "0.000" followed by up and down arrow buttons.

Minimum Per Unit Voltage for	
Constant Power Loads	0.000
Constant Current Loads	0.000

These options are both set to 0 pu during both the base case solution and any contingency case solution.

QV Curves Options: Contingencies

These options are found on the Contingencies sub-tab located on the [Options](#) page of the [QV Curves dialog](#).



Solution Contingencies Output

The QV curve utility allows you to compute QV curves for each of the buses you have selected for the base case and for any number of contingencies. The contingencies you want to process must have already been defined using Simulator's Contingency Analysis feature. Use the checkbox below to specify whether you want to compute QV curves for the presently defined contingencies. If you leave this box unchecked, QV curves will be computed only for base case conditions.

☐ Process each of the currently defined contingencies

☐ Skip base case

☒ Attempt to make unsolvable contingencies solvable with synchronous condenser

Process each of the currently defined contingencies

Check this box to have contingencies included in the QV analysis.

QV curves can be calculated for the specified buses for both base and contingency conditions. To analyze a set of contingencies, these must be defined using the separate [Contingency Analysis Dialog](#). Any contingency whose **Skip** field is set to *NO* will then be included in the QV analysis.

Another option that is relevant to the QV analysis that must be set for a contingency on the contingency analysis dialog is the **QV Autoplot** field. This field must be set to *YES* in order to have a QV curve plotted automatically for a contingency scenario. More information about the option to plot curves automatically is found on the [Output](#) sub-tab.

Contingencies for inclusion with the QV analysis can also be specified based on the results of the PV analysis if choosing to launch the QV analysis immediately following the PV analysis. More information on launching the QV analysis upon completion of the PV analysis is found on the [QV Setup](#) tab on the [PV Curves dialog](#). Even when specifying contingencies based on the PV analysis, this check box must be checked in order to actually include the contingencies with the QV analysis.

Note: The Robust Solution Process option that can be utilized with the contingency analysis when a contingency solution failure occurs is not applied when a contingency is implemented during the QV analysis. All other options defined with contingency analysis such as contingency-specific solution options and make-up power specifications are used when applying a contingency during the QV analysis.

Skip base case

Check this box to prevent the computation of QV curves for base case conditions.

Attempt to make unsolvable contingencies solvable with synchronous condenser

Some contingencies may not solve when implemented. If this box is checked, an attempt will be made to make an unsolvable contingency solvable by providing additional var injection/absorption through use of a fictitious generator (synchronous condenser). As each selected bus is processed,

the fake generator (synchronous condenser) that is normally added to trace the QV curve at a bus, will first be used to determine if Mvar injection/absorption can be adjusted such that a solvable point is found. This process can be quite time consuming. The process first tries to find a solvable point by increasing the studied setpoint voltage and tracing up the curve up to the maximum voltage. It is more likely that a solvable point will be found tracing up the curve due to the increased Mvar injection required to meet the setpoint voltage. If a solvable point is found, this solvable point with the fake generator injection/absorption in place will then be the starting point for the QV curve tracing.

The **Q0** results field found on the [Listing](#) sub-tab of the [Results](#) tab indicates the injection of the fake generator at the base case or initial contingency solution point. This field being non-zero is indicative of an unsolvable contingency with the value of the field being the Mvar injection required to solve the contingency.

Finding a solvable point can be time consuming if there are a number of contingencies that do not solve and a large number of buses are selected for analysis. Uncheck this box if long solution times become an issue.

QV Curves Options: Output

These options are found on the Output sub-tab located on the [Options](#) page of the [QV Curves dialog](#).

Solution Contingencies Output

Save results in comma-delimited format

☒ Save Results to File

Name the file to which to save a comma-delimited version of the results. This will allow you to import the results into a spreadsheet easily. You will NOT be able to import the results back into Simulator using this format. To write a file of results that can later be imported back into Simulator, press the Save button at the bottom of this form and elect to save the results.

Save results in file QVoutput.dat

☐ Plot curves as they are computed

When plotting V versus Q, treat Q as ...

☒ The output of the fictitious synchronous condenser (Qsync)

☐ The total reactive injection at the bus, including shunts but excluding load.

☐ Include reserves (status = closed, generators on AVR, switched shunts not fixed)

Extra Monitoring File

During the analysis, there will be an extra file that gets created for QV results that has little to do with the settings on this page. This is the file used to store the [Quantities to track](#). This file will always be created. There is no option that can be set that will prevent it from being created. By default this file will be named *ExtraMonitoring.csv* and will be stored in the current directory. If a filename is specified in the **Save result in file** box, the extra monitoring file will be named using the filename specified with *_ExtraMonitoring* appended to the filename. The file extension will be the same as that specified with the filename. Regardless of the specified file extension, this file will be a comma-delimited text file.

The results for the tracked quantities will also be stored in an auxiliary file if choosing to store the QV results to file. The extra monitoring file is not meant to replace this but to simply provide a format that is easy to load into Excel for analysis outside of Simulator.

Save results in comma-delimited format

A comma-delimited file format for QV results exists for easy import into a spreadsheet. This file format cannot be loaded back into Simulator. If this functionality is required, it is suggested that the results be saved in an auxiliary file. To save the results to an auxiliary file, click the **Save...** button found at the bottom of the QV Curves dialog. The values saved to the comma-delimited file are the same values that are saved in the SUBDATA section for QVCurve data saved to an auxiliary file.

The comma-delimited file is grouped into sections for each scenario. Each scenario is identified by a header that indicates the bus and either base case or contingency identifier. The header is followed by a line that provides the field order of the actual data. The field line is followed by lines of data. Each scenario grouping is separated by a blank line. All results are sorted by the voltage setpoint.

The fields have the following meaning:

V(PU) - per-unit voltage set-point

Q_sync(MVR) - output of the fake generator (synchronous condenser)

Q_shunt(MVR) - any existing on-line switched shunt output

Q_tot(MVR) - $Q_sync + Q_shunt$

Q_res(MVR) - available reserve at the bus

Q_tot_res(MVR) - $Q_sync + Q_shunt + Q_res$

Here is an example of the comma-delimited file:

```
**BUS** 3 (Three_138.0),**CASE** L_000001ONE-000003THREEC1
V(PU), Q_sync(MVR), Q_shunt(MVR), Q_tot(MVR), Q_res(MVR), Q_tot_res(MVR)
1.1000, 437.5849, 48.4000, 485.9849, 242.0000, 727.9849
1.0962, 419.8296, 48.0627, 467.8923, 240.3136, 708.2059
1.0862, 374.1186, 47.1898, 421.3083, 235.9489, 657.2572
1.0762, 329.2408, 46.3248, 375.5656, 231.6243, 607.1899
1.0662, 285.1944, 45.4679, 330.6623, 227.3396, 558.0020
1.0562, 241.9775, 44.6190, 286.5965, 223.0950, 509.6915
1.0462, 199.5884, 43.7781, 243.3665, 218.8904, 462.2568
1.0362, 158.0253, 42.9451, 200.9705, 214.7257, 415.6962
1.0262, 117.2866, 42.1202, 159.4069, 210.6011, 370.0079
1.0162, 77.3706, 41.3033, 118.6739, 206.5164, 325.1904
1.0062, 38.2757, 40.4944, 78.7701, 202.4718, 281.2419
0.9962, 0.0000, 39.6934, 39.6934, 198.4671, 238.1605
0.9962, 0.0000, 39.6934, 39.6934, 0.0000, 39.6934
0.9962, 0.0000, 39.6934, 39.6934, -119.0803, -79.3869
0.9862, -37.4572, 38.9005, 1.4433, -116.7015, -115.2582
0.9762, -74.0981, 38.1156, -35.9825, -114.3467, -150.3292
0.9662, -109.9612, 37.3386, -72.6225, -112.0160, -184.6385
0.9562, -144.9682, 36.5697, -108.3985, -109.7092, -218.1077
0.9462, -179.1652, 35.8088, -143.3564, -107.4264, -250.7828
0.9362, -212.5517, 35.0559, -177.4958, -105.1676, -282.6635
```

Save Results to File

Check this box to actually save the results to the file specified by **Save results in file**.

Save results in file

Specify the name and file path of the file to which to save the results. This option will be enabled when the **Save Results to File** option is checked. Click the **Browse** button to open a dialog that will allow selection of the file path and name.

Plot curves as they are computed

Check this box to draw and update QV curve plots during each step of the QV analysis process. In order to have QV curves automatically plotted for a contingency during each step of the analysis, the **QV Autoplot** field belonging to that contingency must be set to *YES* on the [Contingency Analysis Dialog](#).

If curves are not plotted at they are computed, they can be easily created upon completion of the analysis from an option found on the [Listing](#) sub-tab on the [Results](#) tab.

When plotting V versus Q, treat Q as...

There are several options that determine how the Q quantity should be plotted on a QV curve. These options are applicable whether plotting the curves automatically as they are computed or plotting the curves upon completion of the analysis.

An example plot is shown below with examples of the plots that result from the four possible options for how to treat Q.

The output of the fictitious synchronous condenser (Qsync)

Select this option to treat Q as the output of the fake generator (synchronous condenser) only. This is the Q_sync(MVR) curve shown in the example plot below.

The total reactive injection at the bus, including shunts but excluding load

Select this option to treat Q as the sum of the output of the fake generator (synchronous condenser) and any existing on-line shunts at the bus. Shunts include both bus shunts and switched shunts. This is the Q_tot(MVR) curve shown in the example plot below.

Include reserves (status = closed, generators on AVR, switched shunts not fixed)

Check this box to offset the Q values by the appropriate Mvar reserves values.

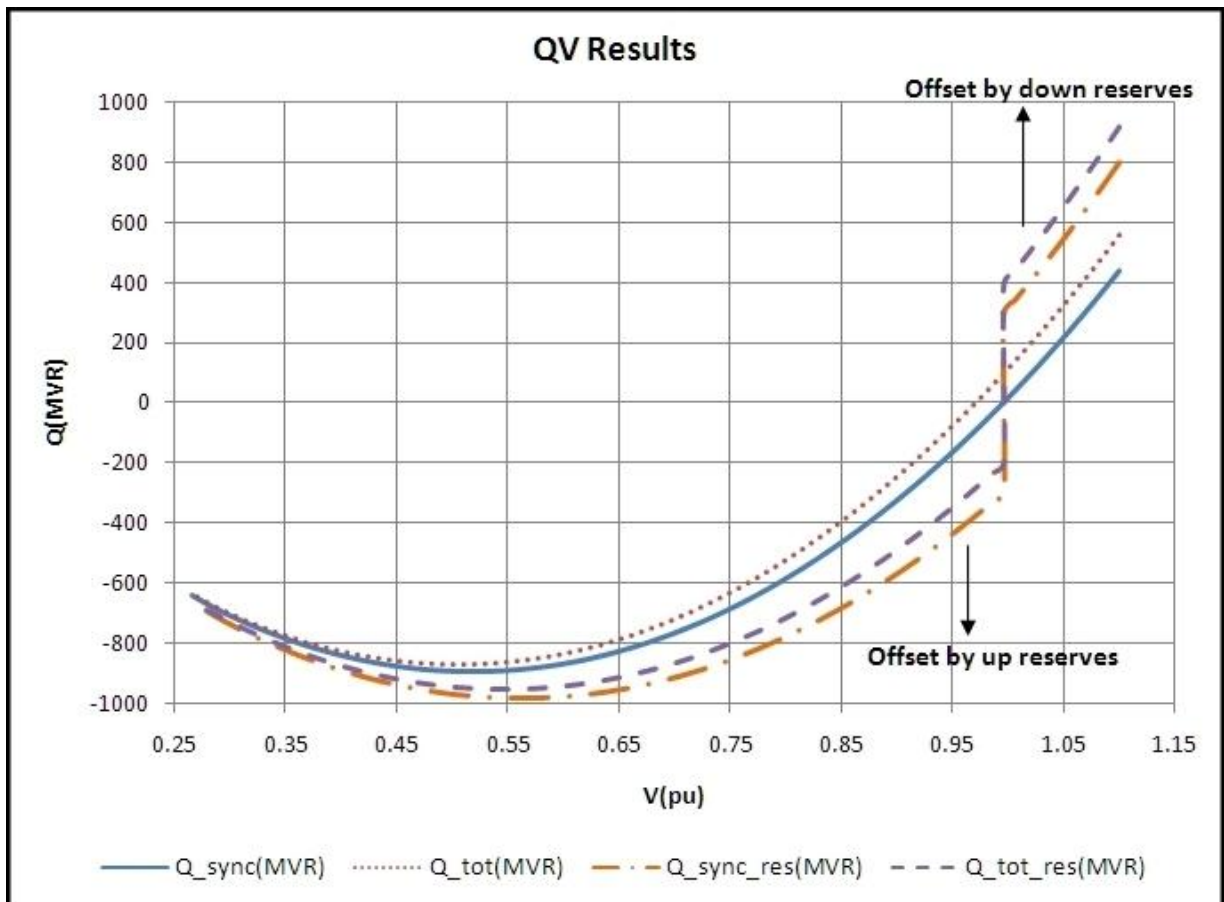
When the QV curve is traced for a given bus, any existing switched shunts on control at that bus are turned off control and any existing generators on AVR control at that bus are turned off control. The amount of Mvar support that these devices can provide is then recorded and determined at each voltage set-point. These are the reserves values. Only devices that are on-line and on control will be included. This means that for a generator the AVR status = *YES* and for a switched shunt the control mode must not be *Fixed*.

The amount of down (decrease) reserves at a given voltage set-point for a switched shunt is determined by the difference between the current nominal output of the shunt and its minimum nominal output multiplied by the square of the voltage set-point: $(\text{Nominal Mvar} - \text{Min Nominal Mvar}) * V^2$. The amount of down (decrease) reserves for a generator at a given voltage set-point is the difference between the current output and the minimum Mvar output: $(\text{Mvar Output} - \text{Min Mvars})$. The amount of up (increase) reserves at a given voltage set-point for a switched shunt is determined by the difference between the maximum nominal output of the shunt and its current nominal output multiplied by the square of the voltage set-point: $(\text{Max Nominal Mvar} - \text{Nominal Mvar}) * V^2$. The amount of up (increase) reserves for a generator at a given voltage set-point is the difference between the maximum Mvar output and the current output: $(\text{Max Mvars} - \text{Mvar Output})$. The total reserves of each type at any voltage set-point is the sum of reserves of that type for all qualifying switched shunts and generators at the studied bus.

Which type (up or down) of reserves to consider at a particular voltage set-point is determined based on the location of the voltage set-point relative to the base case operating voltage. When considering contingencies, the base case operating voltage is the voltage at a bus following implementation of the contingency but before any QV curve tracing is started. If the voltage set-point is less than the base case operating voltage, the up reserves are considered. This part of the curve represents an increase in the Mvar load at a bus. The difference in the bottom of the curve and the x-axis is the amount that the Mvar load at a bus can be increased before voltage collapse. Being able to inject additional Mvar through available reserves will further increase this margin. When plotting including reserves, the portion of the curve to the left of the base case operating point is then offset downward by the amount of up reserves that are available at each voltage set-point, thus increasing the margin before collapse. The portion of the curve to the right of the base case operating point is then offset upward based on the amount of down reserves available at each voltage set-point.

Instead of listing an absolute value of reserves available, the $Q_{\text{res}}(\text{MVR})$ value as reported with the QV output results (either comma-delimited file or auxiliary file) is reflective of the offset required when plotting the curve including the reserves.

When choosing to include reserves, the values that are plotted are either $Q_{\text{sync_res}}(\text{MVR})$ or $Q_{\text{tot_res}}(\text{MVR})$. The $Q_{\text{sync_res}}(\text{MVR})$ value is not included in the QV output results (either comma-delimited file or auxiliary file) but is equal to the sum of $Q_{\text{sync}}(\text{MVR})$ and $Q_{\text{res}}(\text{MVR})$: $Q_{\text{sync_res}}(\text{MVR}) = Q_{\text{sync}}(\text{MVR}) + Q_{\text{res}}(\text{MVR})$. $Q_{\text{sync_res}}(\text{MVR})$ represents the combination of selecting the option to plot **The output of the synchronous condenser** and including reserves. $Q_{\text{tot_res}}(\text{MVR})$ represents the combination of selecting the option to plot **The total reactive injection at the bus...** and including reserves.



Note: The reserves and the existing injection in the above plot are represented solely by switched shunts. All of the curves tend to converge toward the same Q at lower voltages. This is because of the relationship of the actual output of a switched shunt to the nominal output: $(\text{Actual Mvar}) = (\text{Nominal Mvar}) \cdot V^2$.

PV/QV Quantities to Track

Specification of quantities to track during analysis is almost identical for both the PV and QV analysis (exceptions will be noted below). For PV analysis, the Quantities to Track tab is accessed from the [PV Curves](#) dialog, and for the QV analysis, this tab is accessed from the [QV Curves](#) dialog. Quantities to track are monitored and stored as the transfer increases for the PV analysis or the voltage set-point changes for the QV analysis. Any unselected system parameters will not be saved. The Quantities to Track page contains several sub-tabs that allow the monitoring of different types of objects including: buses, generators, injection groups, lines, transformers, shunts, interfaces, and the case as a whole. There is also a sub-tab for monitoring devices at limits.

Each of the object sub-tabs contains fields that can be tracked during the PV analysis. The default tracking for all of these fields is *NO*. To monitor a particular quantity, double-click the corresponding field entry to toggle it to *YES*, or in the case of some line quantities, the appropriate monitoring direction.

Quantities that are tracked can be plotted using the [Plot](#) page. They can also be stored to an auxiliary file with both the PV and QV tools. With the QV analysis a special file is created that stores the tracked quantities. For more information on this see the Extra Monitoring File section with the [QV Curves Options: Output](#) topic.

Buses

Voltage - per unit voltage

kV Voltage - actual voltage in kV

Angle - voltage angle in degrees

MW Load - total real power load

Mvar Load - total reactive power load

dV/dQ - sensitivity of voltage to change in reactive power injection at the same bus

VP Sensitivity - sensitivity of voltage change due to real power transfer. For PV analysis this is the sensitivity of the voltage at the bus due to the selected transfer from source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

VP sensitivities can also be calculated using the [Flow and Voltage Sensitivities tool](#). With this tool there is an option to turn AVR control off for generators and continuous switched shunts that are at buses participating in the transfer. This same option is not available when calculating these sensitivities during the PV analysis because of the overhead of additional power flow solutions that would be required at each transfer step to implement this option. Leaving devices on AVR control will maintain the voltage setpoint at the regulated buses creating PV buses in the power flow Jacobian. At PV buses the sensitivity of voltage to a transfer will be zero.

Shunt Inj - total switched shunt reactive power (Mvar) output

For PV analysis if the intent is to trace PV curves, it makes sense to monitor at least one bus voltage. The process will not automatically monitor and store voltage quantities.

For QV analysis, per unit voltage will be monitored automatically for all buses selected to be included in the QV analysis.

If the [Integrated Topology Processing](#) add-on is available, an additional option will be available on the Buses tracking tab, the **Modify Existing Bus Tracking to Track Only Single Bus Per Super Bus** button. When using topology processing, only a single bus per super bus will be in the consolidated case. That means that the parameters for each bus in the super bus will be the same, i.e. same voltage magnitude and angle, etc. This button can be used to eliminate any excess

tracking that would just take up computer memory by tracking essentially the same quantities at the same bus multiple times. To take advantage of this option and only track a single bus per super bus, first set up the bus tracking to track all of the necessary quantities. Then push the **Modify...** button. The field entries in the table will then be updated to only include a single bus per super bus. When determining the primary node (the primary node is the bus that will ultimately be tracked during the analysis) and super buses, the active defined contingencies will be used to determine how the case is consolidated. Because the consolidation can be different based on the contingency set, or lack thereof, it is best to just define the tracking to include all necessary buses and use this option to refine the list down to a single bus per super bus rather than manually trying to maintain such a list.

Generators

Gen MW - real power output

Gen Mvar - reactive power output

Mvar Reserve - reactive power reserves in the positive direction (Max Mvar - Mvar)

Groups

This allows the monitoring of injection group parameters.

Mvar Reserve - total reactive power reserves in the positive direction for all generators in the group (Max Mvar - Mvar)

Gen MW - total real power output of all generators in the group

Gen Mvar - total reactive power output of all generators in the group

Load MW - total real power load for all loads in the group

Load Mvar - total reactive power load for all loads in the group

Lines

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab.

Branch MW - real power flow on the branch in either the FROM-TO or TO-FROM direction

Branch Mvar - reactive power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MVA - total power flow on the branch in either the FROM-TO or TO-FROM direction

Branch MW Loss - real power loss

Branch Mvar Loss - reactive power loss

Branch PTDF - [Power Transfer Distribution Factor](#) on the branch due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Xfmr Tap - tap ratio if the branch is a transformer

For the flow (Branch MW, Branch Mvar, and Branch MVA) and Branch PTDF fields, double-clicking on a particular entry will toggle its value from *NO* to *FROM-TO*, and double-clicking again will toggle its value to *TO-FROM*. Double-clicking on a loss field or Xfmr Tap field will toggle the value between *YES* and *NO*.

Transformers

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. See **Lines** above for setting up common branch quantities. Only branches containing transformers will appear on the **Transformers** sub-tab.

Type - set the transformer type (Fixed, LTC, Mvar, or Phase)

Reg Val - regulated value for the transformer control

Tap Pos - tap position

Reg Err - deviation of the regulated value from the regulation Min/Max range

All fields specified above except for the Type field are used to determine which quantities should be monitored and stored during the analysis. The Type field can be accessed here in order to change the type of transformer. The type will not be monitored during the analysis.

Shunts

This allows the monitoring of switched shunt parameters.

Actual Q - actual reactive power output

Actual P - actual real power output

Nom Q - nominal reactive power output

Nom P - nominal real power output

Reg Err - deviation of the regulated value from the regulation Min/Max range. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Reg Val - value of regulated quantity at the regulated bus. This can either be in per unit voltage or Mvar depending on the type of regulation that the shunt is doing.

Interfaces

MW Flow - real power flow

Mvar Flow - reactive power flow

MVA Flow - total power flow

MW Loss - real power losses

Mvar Loss - reactive power losses

Interface PTDF - [Power Transfer Distribution Factor](#) on the interface due to a defined real power transfer. This parameter only makes sense when doing the PV analysis. The PTDF is calculated for the transfer from the selected source to sink. This is a meaningless quantity for the QV analysis and should not be monitored.

Case

These options allow the tracking of case quantities to provide a better picture of what is going on in the case as a whole without requiring the individual tracking of quantities that make up the case total. These quantities are determined in the same manner as those found in the Case Totals section of the [Case Summary dialog](#).

Available quantities for tracking include:

Load MW/Mvar - total real or reactive power load for the entire case

Generation MW/Mvar - total real or reactive power generation for the entire case

Shunts MW/Mvar - total real or reactive shunt injection from switched shunts, bus shunts, and line shunts. A positive value will indicate shunt load where a negative value will indicate shunt compensation such as capacitance.

Losses MW/Mvar - total real or reactive power losses for the entire case

ATC Extra Monitors

This option is only available when using the PV Curves tool. The following will appear at the bottom of the Quantities to Track tab accessed from the PV Curves dialog.



[ATC Extra Monitors](#) are typically used with the [ATC add-on tool](#) to monitor special branches or interfaces in addition to ones monitored by setting the [Limit Monitoring Settings](#). This would be done for example if trying to determine an interface flow limit so that interface can be used as a proxy for limiting the transfer between a source and a sink. This same functionality is desirable when using the PV analysis. Both the ATC and PV analyses are similar in that they study the impacts of real power transfers. The ATC analysis is concerned with thermal limitations while the PV analysis looks at voltage stability limitations. Use of the ATC Extra Monitors with the PV analysis could be additionally useful for developing a proxy interface to help prevent voltage stability problems.

Click the **Define ATC Extra Monitors** button to view a list of existing ATC Extra Monitors or create new ones. To include monitoring of these elements at part of the PV analysis, check the **Include ATC Extra Monitors** box.

When monitoring ATC Extra Monitors with the PV analysis, the real power (MW) flow on the specified branches and/or interfaces will be monitored at each base case transfer level. No monitoring is done with contingencies implemented.

ATC Extra Monitors can be added to the [Overview](#) table, available on the [PV Results](#) tab, via the [Display/Column Options...](#) dialog. When displayed in this table, the value reported with the extra monitor is the flow on the extra monitor at the Max Shift level of transfer for the given scenario. Values for ATC Extra Monitors will only be displayed in the Overview table for critical scenarios.

Values for ATC Extra Monitors can also be displayed in the Overview table for transfer levels at which inadequate voltages are found. These values will only be available if using the option found on the [Limit Violations](#) tab to **Store inadequate voltages**. Fields for reporting these values are found in the Inadequate Voltage folder in the list of available fields for the Overview table. The value reported for each extra monitor is the flow on the extra monitor at the **Inadequate Voltage Nominal Shift** transfer level.

Access to the ATC Extra Monitor values stored at transfer levels other than the Max Shift transfer for critical scenarios and values stored for non-critical scenarios is available through SUBDATA sections of the PWPVResultListContainer data type that is saved when saving PV results to an auxiliary file or in the output file saved when choosing to save results to file from the [PV output](#) tab.

Devices at Limits

The **Devices at Limits** sub-tab allows the selection of various options to track devices that hit or back off limits during the PV analysis. Generators and switched shunts at var limits, LTC transformers at tap limits, and lines and interfaces at thermal limits can all be tracked.

Track Hitting and Backing Off Limits during Ramping

<input type="checkbox"/> Generators Var Limits	<input type="text"/>	Define...
<input type="checkbox"/> Switched Shunts Var Limits	<input type="text"/>	Define...
<input type="checkbox"/> LTC Transformers Tap Limits	<input type="text"/>	Define...
<input type="checkbox"/> Lines Thermal Limits	<input type="text"/>	Define...
<input type="checkbox"/> Interfaces Thermal Limits	<input type="text"/>	Define...

Note: Narrow limit tracking by specifying a filter

Note: If no filter is specified, then all objects will be tracked.

To track the limits of any of these devices, check the appropriate checkbox. Next to each checkbox there is a dropdown box for selecting an [advanced filter](#) to apply to the tracking. A new filter can be defined for a particular device by clicking the **Define** button next to that device. The filter will limit the number of devices that get tracked. It is a good idea to define a filter for tracking devices so that all elements in the case will not be tracked. Keep in mind that tracking any quantities will take up space in the computer memory. If too many devices are tracked, it is possible to run out of memory.

With these options, device limits are only tracked in the base case at valid transfer levels. No device limit tracking is done with contingencies implemented.

QV Curves: Results

The Results tab is found on the [QV Curves dialog](#). This page provides access to all of the QV results. This page is divided into several additional sub-tabs from which different categories of results are available:

[Listing](#)

[Plots](#)

[Track Limits](#)

The Results tab also displays messages about the progress of the analysis. The messages are displayed regardless of the sub-tab that is selected.

QV Curves Results: Listing

The Listing sub-tab is found on the [Results](#) page of the [QV Curves dialog](#). This page provides a summary of all scenarios studied and their results. This table will update as the analysis progresses.

Listing	Plots	Track Limits								
	Number	Name	Nom kV	Case Name	V at Q0	Q0	Qinj_0	Vmax	Q at VMax	Q
1	5 Five	138.00	BASECASE		0.95	0.00	0.00	1.1000	191.52	
2	5 Five	138.00	L_000001ONE-000002TWOC1		0.96	0.00	0.00	1.1000	191.14	
3	5 Five	138.00	L_000001ONE-000003THREEC1		0.95	0.00	0.00	1.1000	192.15	
4	5 Five	138.00	L_000002TWO-000003THREEC1		0.95	0.00	0.00	1.1000	192.23	
5	5 Five	138.00	L_000002TWO-000004FOURC1		0.95	0.00	0.00	1.1000	192.31	
6	5 Five	138.00	L_000002TWO-000005FIVEC1		0.95	91.67	91.67	1.1000	159.12	
7	5 Five	138.00	L_000002TWO-000006SIXC1		0.00	0.00	0.00	0.0000	0.00	
8	5 Five	138.00	L_000003THREE-000004FOURC1		0.96	0.00	0.00	1.1000	191.12	
9	5 Five	138.00	L_000004FOUR-000005FIVEC1		0.92	0.00	0.00	1.1000	151.38	
10	5 Five	138.00	L_000007SEVEN-000005FIVEC1		0.95	0.00	0.00	1.1000	191.51	
11	5 Five	138.00	L_000006SIX-000007SEVENC1		0.95	0.00	0.00	1.1000	191.51	
12	5 Five	138.00	L_000006SIX-000007SEVENC2		0.95	0.00	0.00	1.1000	191.51	

The listing is a [case information display](#) and therefore can be used in a manner consistent with all other case information displays.

There are a number of fields that are shown by default on the display:

Number, Name, Nom kV

Number, name, and nominal kV voltage of the studied bus, respectively. This along with the Case Name identifies the scenario.

Case Name

Either *BASECASE* or the name of a contingency. This along with the **Number**, **Name**, and **Nom kV** identifies the scenario.

V at Q0

Per-unit voltage in the starting case (base case operating point). The starting case is either the base case or a contingency case. This is the voltage that is recorded prior to starting any QV curve tracing. This is the voltage at **Q0**.

If the case is not solvable and a successful attempt is made at finding the solvable point by using the fake generator to provide Mvar injection, this is the voltage recorded once the case becomes solvable.

Q0

Mvar output of the fake generator in the starting case (base case operating point). The starting case is either the base case or contingency case. This is the Mvar output that is recorded prior to starting any QV curve tracing.

If the case is not solvable and a successful attempt is made at finding the solvable point by using the fake generator to provide Mvar injection, this is the Mvar output of the fake generator that is

required to make the case solvable. Therefore, a non-zero entry means that the initial starting case was not solvable.

Qinj_0

Total Mvar injection in the starting case (base case operating point). The starting case is either the base case or contingency case. This is the total injection that is recorded prior to starting any QV curve tracing. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

Vmax

Maximum per-unit voltage that was recorded.

Q at VMax

Mvar injection of the fake generator at the maximum per-unit voltage that was recorded, **Vmax**.

Qinj at Vmax

Total Mvar injection at the maximum per-unit voltage that was recorded, **Vmax**. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

V at Qmin

Per-unit voltage at the minimum recorded fake generator injection, **Qmin**.

Qmin

Minimum recorded fake generator injection. If negative, this is the amount that the Mvar load can be increased before voltage collapse. If positive, this is the amount of Mvar injection required to come out of collapse and achieve a solvable case.

Qinj_min

Total minimum recorded Mvar injection. This includes the shunt injection from bus shunts and switched shunts and the output of the fake generator.

Vmin

Minimum per-unit voltage that was recorded.

Q at Vmin

Mvar injection of the fake generator at the minimum per-unit voltage that was recorded, **Vmin**.

Qinj at Vmin

Total Mvar injection at the minimum per-unit voltage that was recorded, **Vmin**. This includes the shunt injection from bus shunts and switched shunt and the output of the fake generator.

The right-click local menu has one option specific to QV results:

Plot QV Curve

This will plot the QV curve for the selected scenario. What value of Q to plot is determined by the options selected for how to treat Q as found on the [Output](#) sub-tab of the [Options](#) page. If multiple scenarios are selected, the curves for the multiple scenarios will be shown on the same plot.

PV/QV Curves Results: Plot

Plotting functionality is almost identical for both the PV analysis and QV analysis (exceptions will be noted below). For PV analysis, the Plot tab is accessed from the [PV Results](#) tab of the [PV Curves dialog](#). For QV analysis, the Plots tab is accessed from the [QV Results](#) tab of the [QV Curves dialog](#). Any of the values selected to be [monitored](#) can be plotted using this display.

Horizontal axis value ☐ Pre-contingency
Nominal Shift

Vertical axis value type
PU Volt

Plot values for these elements ...

Description	Plot?
Three (3)	NO
Five (5)	NO

For these scenarios ... ☐ Always include base case

PV Scenario	Critical?	Plot?
base case	NO	NO
L_000001One-000002TwoC1	YES	NO
L_000001One-000003ThreeC1	NO	NO
L_000002Two-000003ThreeC1	NO	NO
L_000002Two-000004FourC1	NO	NO
L_000002Two-000005FiveC1	NO	NO
L_000002Two-000006SixC1	YES	NO
L_000003Three-000004FourC1	NO	NO
L_000004Four-000005FiveC1	NO	NO
L_000007Seven-000005FiveC1	YES	NO
L_000006Six-000007SevenC1	YES	NO

Plot title

Clear Plot

The page is broken into two basic sections: the left and right half. The right half of the display allows selection of the scenarios that should be included on a given plot. The left half allows selection of the quantities that will be charted on the plot.

Selecting Scenarios to Plot

The right half of the display gives a table of all scenarios that were studied. Change the **Plot?** field to **YES** to include a scenario in the current plot. The Plot? field is a toggleable field and can be changed from **YES** to **NO** and back again by double-clicking on the appropriate entry. Any scenario whose Plot? field is set to **YES** will be included on the same plot once the **Plot** button is clicked.

The **Always include base case** box should be checked if the base case scenario should be included in the plot along with any other scenarios selected. This option is not available for QV results plotting.

Before a plot can be created, the quantities to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Selecting Quantities to Plot

The left half of the display gives the various options for selecting what quantities should be plotted.

The **Horizontal axis value** drop-down contains any parameters that were monitored. Additionally, for the PV analysis, this drop-down also contains the independent quantities (Nominal Shift, Export, and Import) recorded during the analysis. If the **Pre-contingency** box is checked, the x values for each plotted scenario will come from the base case instead of the results of a given scenario. The Pre-contingency check box is not available for QV results plotting.

The **Vertical axis value type** drop-down contains a list of all the types of values that were monitored. The type selected in this drop-down will determine the actual elements that are listed in the **Plot values for these elements** table. To include an element in the plot, change the **Plot?** field to **YES** for that element. The Plot? field is a toggleable field and can be changed from **YES** to

NO and back again by double-clicking on the appropriate entry. Any element whose Plot? field is set to *YES* will be included in the same plot once the **Plot** button is clicked.

Before a plot can be created, the scenarios to plot must also be set. The results for each selected element will be plotted for each of the selected scenarios.

Plot Title

Optional field that can be set to add a title to a plot and distinguish one plot from another.

Plot

After selecting the scenarios and values to plot, click this button to actually generate the plot. The plot will appear in its own window. The number of plots that can be created is limited only by computer system memory.

Right-clicking on a plot will expose a local menu with several options. The plot can be saved as a bitmap, Windows metafile, JPEG, or text file by clicking **Save** and selecting the appropriate file type. **Copy** will allow the plot to be copied and pasted into other applications. **Print** will send the plot to a printer that can be configured by selecting the **Printer Setup** option. Selecting **Close** will close the plot window.

Clear

Click this button to reset all of the **Plot?** fields to *NO* for both the scenarios to plot and the values to plot.

PV/QV Refine Model

Modeling idiosyncrasies may exist that cause premature loss of convergence during either a PV or QV analysis. The PVQV add-on has a tool to modify the currently open case in an attempt to fix some of these idiosyncrasies. This is the Refine Model tool that is available on the [Add Ons](#) ribbon tab in the **PV and QV Curves (PVQV)** ribbon group.

Apply to		
Areas		
#	Name	Apply?
1	Top	NO
2	Left	NO
3	Right	NO

The case can be refined with the following options:

Fix transformer taps

If there are transformers that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all transformer taps at their present values if their Vmax - Vmin is less than or equal to the user specified tolerance.

Fix shunts

If there are switched shunts that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all switched shunts at their present values if their Vmax-Vmin is less than or equal to the user specified tolerance.

Take units off AVR control

If there are generators that have Qmax and Qmin that are very close together, the power flow may have a difficult time converging. This option allows the user to remove these units from AVR control, thus locking their MVAR output at its present value, if their Qmax - Qmin is less than or equal to the user specified tolerance.

Apply to

These refinements will only be applied to those areas or zones that have the **Apply?** field set as **YES** in this dialog box. This field can be toggled between **YES** and **NO** by left-clicking on the appropriate entry.

OK

Click this button to make the selected changes to the currently open power flow case. After the changes are made the **Close** button must be clicked to close the dialog. If any changes have been made and the dialog is closed, the user will be prompted to save the case to a file.

Close

Click this button to close the dialog without implementing any changes to the power flow case in memory.

PowerWorld Simulator Optimal Power Flow Overview

Note: The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package. To learn more about the OPF, please feel free to read through the information contained in these help files. [Contact PowerWorld Corporation](#) for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However with the optimal power flow (OPF) enhancement, Simulator OPF can also solve these equations using an OPF. In particular, Simulator OPF uses a linear programming (LP) OPF implementation.

All of the OPF commands and options are accessed using the LP OPF main menu item. Other commands in this menu are used to specify input options, see results, and store/retrieve OPF specific data into auxiliary files.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#) taking into account both [equality and inequality constraints](#) which are used to model the power balance constraints and various operating limits.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.

OPF Objective Function

The objective of the OPF algorithm is to minimize the OPF objective function, subject to various equality and inequality constraints. Since the objective of the OPF is to minimize an objective function, what objective function is used has a significant impact on the final solution.

Currently two objective functions are available in Simulator OPF: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute values of the changes in the generation in the specified areas or super areas.

To include an area or super area in the OPF objective function, simply change the Area AGC Status field to "OPF" on the [OPF Area Records Display](#) or the Super Area AGC Status field to "OPF" on [the OPF Super Area Records Display](#). This gives you great flexibility in defining the OPF study. For example you can set the OPF to minimize costs for the entire system, or just selected areas or super areas.

OPF Equality and Inequality Constraints

In solving a constrained optimization problem, such as the OPF, there are two general classes of constraints, equality and inequality. Equality constraints are constraints that always have to be enforced. That is, they are always "binding". For example in the OPF the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance); likewise the area MW interchange constraints. In contrast, inequality constraints may or may not be binding. For example, a line MVA flow may or may not be at its limit, or a generator real power output may or may not be at its maximum limit.

An important point to note is because the OPF is solved by iterating between a power flow solution and an LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiate the LP OPF from a standard power flow are the constraints that are explicitly enforced by the LP. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)

OPF Equality Constraints

Area MW Interchange

The area MW interchange constraints are enforced during the LP for those areas that have an **AGC Status** equal to *OPF* provided the area is not part of a super area that is also set on AGC. The **AGC Status** field for an area can be set using the [OPF Area Records display](#), while the **AGC Status** field for the super area (if any) is set using the [OPF Super Area Records display](#). Areas whose interchange is enforced during the LP do not have their interchange enforced during the power flow solution that is done as part of the LP process; during the power flow these areas are treated as though they were off of AGC (and hence the output of generators in that area is not varied during the power flow).

It is perfectly acceptable to have some areas on *OPF* AGC control and to have other areas on the more traditional power flow area AGC such as *ED* or *Part. AGC*. The interchange for such areas is controlled during the power flow solution.

Following a successful solution, marginal costs are calculated for the area interchange constraints; these values are displayed on the [OPF Area Records display](#) and can be contoured. See [OPF Marginal Costs](#) for details.

Bus MW and Mvar power balance

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the bus MW (real power) balance constraint; these values are displayed on the [OPF Bus Records display](#) and can be contoured.

Generator Voltage Setpoint

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the voltage setpoint constraint; these values are displayed on the [OPF Bus Records display](#).

Super Area MW Interchange

Super area interchange constraints are enforced similar to the area constraints. That is, super area interchange constraints are enforced during the LP only for those super areas that have an **AGC Status** equal to *OPF*. The **AGC Status** field can be set using [the OPF Super Area Records display](#). During the power flow solution that is part of the LP process, such super areas are treated as though they were off of AGC.

Interface MW limits when treated as Equality

Interface MW limits are enforced during the LP solution. Interface MW limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Interface Dialog](#) for information on how to treat the limit as an equality constraint.

Transmission Line and Transformer (Branch) MVA limits

Branch MVA limits are enforced during the LP solution. Branch MVA limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Branch Information Dialog](#) for information on how to treat the limit as an equality constraint.

OPF Inequality Constraints

The following classes of inequality constraints are enforced during the OPF solution.

Generator real power limits

Generator real power limits are enforced during the LP solution.

Generator reactive power limits

Generator reactive power limits are enforced during the power flow solution.

Interface MW limits

Interface MW limits are enforced during the LP solution. In short, interface records are used to represent the aggregate flow through a number of different devices (see [Interface Records](#) for details). During the LP the MW flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Options and Results dialog](#) or the [OPF Interface Records display](#). The default is that case interface enforcement is not disabled. Also note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from [the OPF Area Records display](#). Note: the default is that interface enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be set from [the OPF Interface Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Interface Records display](#).

Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the LP solution. During the LP the branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF and Results Options dialog](#) or the [OPF Line/Transformer Records display](#). The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. For tie-lines enforcement must be active for either area. This field can be set from the [OPF Line/Transformer Records display](#). The default is that branch enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Line/Transformer Records display](#).

Determining Set of Active Inequality Constraints

A key issue in quickly solving the OPF is for the LP to effectively determine the set of active inequality constraints. Currently this includes the line MVA limits and the interface MW limits. Because the speed of the LP varies as the cube of the number of constraints active in the LP basis, it is extremely important to keep this number as small as possible. Therefore it would be very computationally prohibitive to setup an inequality constraint for each transmission line and interface (except in very small systems.)

The solution of setting up constraints **only** for those inequality constraints that are actually violating their limits is a step in the right direction, but suffers from the problem that during a solution a line may initially be violating its limit and then after the first iteration it is no longer violating. However if it is not subsequently included as a constraint during the next iteration the solution may simply oscillate between enforcing/unenforcing this constraint. This problem can be resolved by keeping that constraint in the basis even though it is no longer binding.

However this raises a question about how to handle these constraints during future OPF solutions. For example what would happen if a user solved the OPF, and then immediately resolved the OPF. Following the first solution the constraint would be enforced so that it may actually be less than its limit. However if this constraint is not included in the LP basis during the next solution the constraint may immediately violate during the first iteration, requiring a number of iterations just to return to the original initial solution.

Simulator solves this issue by keeping track of the enforced constraints from one solution to the next. Constraints are only removed from the basis if they fall below a specified percentage of their limit. This percentage is enterable on the [Constraint Options page](#) of the [OPF Options and Results Dialog](#). This prevents the set of constraints in the basis from building up over time as a number of different system conditions (and hence constraints) are studied. Also, at any time this set of constraints can be cleared using the **Initialize OPF Button** on the [OPF Options and Results Dialog](#).

Also, the user is free to specify that a particular constraint **always** be included in the basis. This is done by toggling the **Constraint** field to "Always" on the [OPF Line/Transformer Records](#) or [OPF Interface Records](#) displays.

OPF Unenforceable Constraints

The goal of the LP OPF is to minimize the objective function subject to the user specified constraints. However there is no guarantee that it is even possible to simultaneously satisfy all of the specified constraints. In fact, it is quite easy to create a system in which all of the constraints **cannot** be enforced. A simple example is a two bus system consisting of a single generator supplying a single load through a transmission line. If the transmission line MVA rating is below the MVA of the load then it is impossible to supply this load while simultaneously satisfying the transmission line constraint. In Simulator OPF such a situation is known as an unenforceable constraint. In studying large systems, such as the U.S FERC 715 cases, such situations actually appear to be quite common. Seemingly unenforceable constraints are often due to a lack of controls available to the LP OPF or due to faulty limits entered in the case. In such cases unenforceable constraints can be corrected by making more controls available to the LP OPF or correcting the limits.

Simulator OPF allows you to solve systems with unenforceable constraints by only enforcing those constraints that have a marginal cost below a user specified tolerance. These tolerances are specified on the [OPF Options Constraint Options Page](#). Any constraints that have marginal costs above these values are not enforced, including any unenforceable constraints. This functionality is implemented in Simulator OPF through the use of slack variables. Slack variables are artificial variables introduced during the LP solution in order to satisfy the constraints with the slack variable costs equal to the user specified values. Then, during the LP solution the slack variables are usually removed from the LP basis. The only time this does not occur is if the constraint can not be enforced with a marginal cost less than the specified value. The number of unenforceable constraints are shown on the [OPF Option Solution Results Page](#).

OPF Marginal Costs

During any constrained minimization there is a cost associated with enforcing the equality constraints and the binding inequality constraints. These costs are known as the marginal costs.

In Simulator OPF marginal costs are calculated for the following record types:

Bus MW Equality Constraints

The Bus MW marginal costs tell the incremental cost to supply one additional MW of load at the specified bus. These values can be viewed on the [OPF Bus Records display](#); they can also be contoured or viewed on the one-lines using [bus fields](#).

In the absence of any binding inequality constraints (such as Line MVA constraints) all of the bus marginal costs in an area should be identical. Bus marginal costs can only be determined for buses that are in areas or super areas on OPF control.

Area MW Equality Constraints

The Area MW marginal costs tell the incremental cost for the specified area to import one additional MW of load **from the system slack bus**. These values can be viewed on [the OPF Area Records display](#); they can also be contoured or viewed on the one-lines [using area fields](#). In the absence of any binding inequality constraints the area MW marginal cost is identical to the bus MW marginal costs for all the buses in the area. When there are binding inequality constraints this is no longer the case.

Super Area MW Equality Constraints

The Super Area MW marginal costs are identical to the area marginal costs except they apply to super areas rather than areas.

Interface MW Constraints

The Interface MW marginal costs tell the incremental cost of enforcing the interface MW constraints. These values are only nonzero if the interface constraint is actually active (binding); they can be viewed using the [OPF Interface Records display](#).

Line/Transformer MVA Constraints

The Line/Transformer marginal costs tell the incremental cost of enforcing the line or transformer MVA constraint. These values are only nonzero if the line or transformer constraint is actually active; they can be viewed using the [OPF Line/Transformer Records display](#).

OPF Primal LP

Go to the [Add Ons](#) ribbon tab and select **Primal LP** from the **OPF** ribbon group to solve the OPF using the primal LP algorithm.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power flow and then solving a linear program to change the system controls to remove any limit violations. The basic steps in the LP OPF algorithm are

1. Solve the power flow
2. Linearize the power system about the current power flow solution. Both constraints and controls are linearized.
3. Solve the linearly-constrained OPF problem using a primal LP algorithm, computing the incremental change in the control variables. Slack variables are introduced to make the problem initially feasible. That is, the slack variables are used to satisfy the equality and inequality constraints. The slack variables typically have high costs so that during the iteration the slack variables change to satisfy the constraints. The LP then determines the optimal, feasible solution for the linear problem.
4. Update the control variables and resolve the power flow.
5. If the changes in the control variables are below a tolerance then the solution has been reached; otherwise go to step 2.
6. Finish by resolving the power flow.

OPF Future Enhancements

While we certainly plan on introducing additional functionality in future releases, we do want to be as clear as possible about what functionality is not currently provided.

The current version of Simulator OPF allows users to calculate the optimal solution to a power system using generator real power MW outputs and phase shifters as controls, while enforcing area, super area, interface MW and line/transformer MVA constraints. Marginal losses can also be included in the OPF calculation.

Some functionality that is **not** included in the current version of Simulator OPF, and which we hope to include in future versions, include the following:

- Enforcing bus low/high voltage magnitudes as limits
- Including additional devices as controls, such as generator voltage setpoints, LTC transformers, switched shunts.
- Allowing the optimization of different cost functions.
- Additional functionality as suggested by customers.

OPF Options and Results

The OPF Options and Results dialog allows you to customize the OPF solution. To display this dialog, go to the [Add Ons](#) ribbon tab and select **Options and Results** from the **OPF** ribbon group. The dialog consists of three general pages; Options, Results and LP Solution Details.

The Options page has four tabs as well, [Common Options](#), [Constraint Options](#), [Control Options](#), and [Advanced Options](#).

The Results page has four tabs, [Bus MW Marginal Price Details](#), [Bus Mvar Marginal Price Details](#), [Bus Marginal Controls](#), and [Solution Summary](#).

The LP Solutions Page has five tabs, [All LP Variables](#), [LP Basic Variables](#), [LP Basis Matrix](#), [Inverse of LP Basis](#), and [Trace Solution](#).

The dialog also has several buttons at the bottom of the display:

OK, Cancel

Select to close the dialog. Selecting **OK** saves your changes while **Cancel** does not. Note that changes are also saved anytime you select **Solve LP OPF** or **Single Outer Loop**.

Solve LP OPF

Solves the OPF using the Primal LP algorithm. Equivalent to selecting [Primal LP](#) from the **Optimal Power Flow** ribbon group on the [Add Ons](#) ribbon tab.

Single Outer Loop

Does a single outer loop of the Primal LP algorithm.

Initialize LP OPF

Returns the LP OPF variables to their original states.

Print

Prints the selected page of the dialog.

Help

Displays this help page. To view help for a particular page place the cursor on the page and press the F1 key.

OPF Options: Common Options

The [OPF Dialog](#), Common Options page displays general options associated with the OPF solution. The display contains the following fields:

[Objective Function](#)

Allows a choice of solving the LP using either a minimum cost or a minimum control change objective function.

OPF Reserves Options

These options are only available with the [OPF Reserves add-on](#).

Include OPF Reserve Requirements

Check this box to include OPF reserve constraints in the OPF calculation.

Use a Single Control for Up/Down Regulation

Check this box to use a single bidirectional control for regulating reserve up or down. A unit will provide the same amount of regulating reserve up or down.

LP Control Variables

Disable All Phase Shifter Controls

Prevents phase shifters from attempting to control devices during the OPF solution.

Disable All Generator MW Controls

Prevents generators from shifting MW output during the OPF solution.

Disable All Load MW Controls

Prevents loads from shifting MW demand during the OPF solution.

Disable Area-toArea MW Transaction Controls

Prevents MW transactions between areas from being dispatched during the OPF solution.

Disable DC Transmission Line MW Controls

Prevents shifting of DC Transmission Line MW transfers during the OPF solution.

LP Options

Maximum Number of LP Iterations

Maximum number of allowable iterations for the LP portion of the LP OPF. How many iterations are required to obtain a solution depends, among other things, upon the number of breakpoints in the control cost models. Since each LP iteration can only move from one breakpoint to the next, the finer the model the more iterations required. However the LP is quite fast so a large number of iterations can be performed quite quickly. Default = 1000. Select *Treat Maximum LP Iterations Solutions as Valid* will treat the maximum number of iteration as a valid solution in the LP OPF.

Phase Shifter Cost (\$ / Degree)

Specifies the assumed cost for moving phase shifting transformer taps away from their initial values. The purpose for this fictitious cost is approximate the cost of actually changing the angle of a phase

shifting transformer, and to avoid large changes in phase shifter angles that have very little impact on the system. This field may be zero. Default = \$ 0.10 / Degree.

Calculate Bus Marginal Cost of Reactive Power

When this option is checked, the OPF algorithm will also calculate the marginal cost of reactive power at each bus. Typically the result of interest from the OPF algorithm is the MW marginal cost of each bus (the LMP), but the MVAR marginal cost can be determined as well.

Save Full OPF Results in PWB File

When checked, Simulator will store the full set of results, including the LP matrix, in the PowerWorld Binary case file.

Do Detailed LP Logging

When checked, Simulator will write details on the LP algorithm solution during each pivot of the LP matrix. This is useful for debugging LP solution issues when running a LP OPF solution.

Start with Last Valid OPF Solution

When checked, Simulator will start the LP OPF solution process using the most recent OPF solution as the initial conditions.

OPF Options: Constraint Options

The OPF Dialog, Constraint Options page displays options associated with the enforcement of the constraints by the OPF. The display contains the following fields:

Line/Transformer Constraints

Disable Line/Transformer MVA Limit Enforcement

Select to disable enforcement of Line/Transformer MVA constraints for the entire case.

Percent Correction Tolerance

Specifies a tolerance for the enforcement of line/transformer MVA flows. The tolerance is necessary to prevent solution oscillations due to the non-linear nature of the actual constraints.

Violated elements are always enforced to their limits multiplied by the MVA Enforcement Percentage. If power systems were completely linear then following the LP solution the constraint would actually be equal to this value. However because of nonlinearities, the constraint is close to this value but usually not identical to the value. The Percent Correction Tolerance is used to tell the OPF how close is close enough. Provided all the constraints are violating their limits by less than the correction tolerance percentage the optimal solution is assumed to have been found. You may set this value as low as you like, but setting it too close to zero may result in convergence difficulties. The default is 2 percent.

MVA Auto Release Percentage

Specifies a MVA level at which transmission lines can be released as an OPF constraint equation if the branch MVA flow falls below the level specified.

Maximum Violation Cost (\$/MWhr)

If a branch MVA limit cannot be enforced during an OPF solution, the branch will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

Enforce Line/Transformer MW Flow Limits (Not MVA)

Checking this box will cause Simulator to treat the limits of the transmission elements as MW limits instead of MVA limits. Thus Simulator will report violations on these elements in the OPF based on the MW flow of the element versus the element's MVA rating.

Interface Constraints

Disable Interface MW Limit Enforcement

Select to disable enforcement of Interface MW constraints for the entire case.

Percent Correction Tolerance, MW Auto Release Percentage, Maximum Violation Cost (\$/MWhr)

These fields are equivalent to the entries described above for Line/Transformer MVA Constraints except that they apply to Interface MW constraints.

Monitor/Enforce Contingent Interface Limits

This option allows you to choose if contingency elements in interfaces should be enforced during the OPF solution. Even if they are not enforced, the flows on the remaining elements in the interface will be monitored. The choices you have for enforcing contingency elements in interfaces during the OPF are Never, Power Flow/OPF but not CA/SCOPF, or All Applications including CA/SCOPF. The reason

CA ([contingency analysis](#)) and SCOPF ([Security Constrained OPF](#)) are singled out is because those two tools are already looking at contingency actions separately, outside of the interface definitions. Therefore it may be desired to ignore the inclusion of contingency elements within an interface definition when using these two tools.

Phase Shifting Transformer Regulation Limits

Disable Phase Shifter Regulation Limit Enforcement

Select to disable enforcement of Phase Shifter regulation limit constraints for the entire case.

For a phase shifter, there are two limits which can be enforced during the OPF—the MVA branch rating, and the regulation minimum and maximum MW flow. Checking this option will globally disable the enforcement of the regulation minimum and maximum MW flow for all phase shifters.

In Range Cost (\$/degreehr)

The cost of changing a phase shifter angle away from zero degrees in \$/degreehr, when the angle is within the allowable range of the phase shifter.

Maximum Violation Cost (\$/MWhr)

If a phase shifter regulation limit cannot be enforced during an OPF solution, it will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

Limit Monitoring Settings...

This button opens the [Limit Monitoring Settings](#) dialog, which allows you to change the enforcement percentages for monitored elements.

OPF Options: Control Options

The LP OPF Dialog, Control Options page displays options for generator control and power flow solution. The display contains the following options:

Generator Control Options

Allow Commitment of Fast Start Generators

If this option is checked, then generators designated as Fast Start generators can be turned on or "committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

Allow Decommitment of Fast Start Generators

If this option is checked, then generators designated as Fast Start generators can be turned off or "de-committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

Round to On Percentage if Min Limit

Generators designated as Fast Start generators will be turned On when the generator output is at the value specified by this option.

Modeling Generators without Piecewise Linear Cost Curves

The following fields specify how the OPF should handle generators that are specified as having a cubic cost model. Because the OPF is based upon an LP implementation, all control costs must be modeled using piecewise linear cost curves. These options permit an automatic conversion of cubic models to piecewise linear models. Alternatively, you can very easily convert the cubic models manually using the **# Cost Curve Points** field on [the OPF Generator Records display](#) or using the [generator dialog](#).

Generators Cost Models

This field specifies how generators with cubic cost models should be handled in the OPF. The field has three values

Ignore Them - Generators with cubic cost models are Ignored during the OPF solution. That is, they are considered as though their AGC status was off.

Change to Specified Points per Curve - A piecewise linear cost model is automatically inserted for the generator with a fixed number of points specified in the **Total Points Per Cost Curve** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified. This is the default value.

Change to Specified MWs per Segment - A piecewise linear cost model is automatically inserted for the generator such that each segment in the cost model covers the amount of MWs specified in the **MWs per Cost Curve Segment** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified.

Total Points Per Cost Curve

Specifies the total number of segments that should be automatically inserted into the piecewise linear cost models for those generators that are modeled using cubic cost functions. This is only

done if the **Generator Cost Modeling** field is **Change to Specified Points per Curve**. Default = 5.

MWs per Cost Curve Segment

Specifies the number of MWs for each segment of the piecewise linear cost models that are automatically inserted for those generators that are modeled using cubic cost functions. This is only done if the **Generator Cost Modeling** field is **Change to Specified MWs per Segment**. Default = 10 MW.

Save Existing Piecewise Linear Cost Curves

Generators that are modeled with cubic cost curves may have existing piecewise linear cost curves which may have been manually entered by the user. These curves may or may not resemble the cubic cost function. During the OPF solution the existing piecewise linear cost curves are replaced with the auto-created cost curves. If this option is checked then the existing piecewise linear cost curves are restored at the end of the OPF. The default and recommended option is false since this allows one to view the actual cost curves used by the OPF.

If you would like to use a particular piecewise linear cost function simply make sure that the generator is modeled using the piecewise linear model, which can be set on the [OPF Generator Records](#) display.

Modeling of OPF Areas/Superareas

During the Initial OPF Power Flow Solution

Choose what manner of generation control you wish to be employed in the FIRST power flow solution the OPF will perform, which will establish the base case load flow condition for performing the subsequent OPF generation dispatch.

During Stand-Alone Power Flow Solutions

Choose what manner of generation control you wish to be employed in all load flow solutions FOLLOWING the initial load flow solution. In other words, after the LP OPF routine has determined the new generation dispatch, what type generation dispatch should be used during the normal load flow solution.

NOTE: it is NOT recommended that you use Economic Dispatch in this case, although it is an available option. The reason it is not recommended is that you will remove the optimal dispatch (including constraints) just determined by the OPF in favor of lowest cost economic dispatch, which will likely result in the re-introduction of overloaded elements that were corrected by the OPF dispatch in the first place.

Treat Area/Superarea MW Constraints as unenforceable even when the ACE is less than the AGC Tolerance (default is checked)

This option by default is checked, which means that if Simulator solves an OPF and the generators in an area or super area are all either at their minimum or maximum outputs, and the area control error is still not 0, then the area MW constraint is considered unenforceable and Simulator will assign a large penalty cost (usually \$5,000/MW hr) to the LMP's of the area. However, if you deselect this option, you give Simulator the additional control of checking the ACE mismatch against the AGC tolerance for the area, and if the ACE mismatch is less than the AGC tolerance, Simulator will "acquire" the mismatch amount from the system slack bus for the area, and the area will not be considered unenforceable in the OPF solution. The AGC tolerance is usually small, and therefore the amount of power taken from the system slack bus will also be small. The purpose of this option is to allow for valid LMP solutions in an area that is meant to reach its full minimum or maximum

generator capability, but that losses and numerical rounding results in the ACE mismatch deviating slightly from 0. It is not advised to uncheck this option if you have areas with large AGC tolerances.

OPF Options: Advanced Options

Detection of LP Cycling

This option allows for advanced tweaking of when cycling is detected in the LP. It is recommended that users leave the default settings in place.

Minimum Number of Degenerate Iterations to Assume Cycling

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations before cycling is identified can be specified manually in this box.

Minimum Number of Degenerate Iterations Multiplied by Number of Tableau Rows

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations as a multiple of the number of tableau rows can be specified manually in this box.

Number of Sequential Degenerate Iterations for Last Case

This displays the number of degenerate iterations detected when performing the most recent OPF solution.

Power Flow Recalculation

Choose one of the options to determine how often the power flow is resolved. The three options are, "When total generator MW change > than tolerance"; "After each LP solution"; and "Only at end of LP OPF". If the power flow is not resolved, linear sensitivities are used to determine which constraints enter the LP tableau.

Total Generator Change Tolerance (MW)

Specifies the total generator change tolerance. The default is 500 MW.

After each LP solution

Resolves the power flow upon completion of each LP solution

Only at end of LP OPF

Resolves the power flow only after the LP OPF has completed.

OPF Options: Solution Summary

The OPF Dialog, Solution Summary page displays general results from the last OPF solution. Click the **Save as Aux** button to save these results to a Simulator Auxiliary file.

The display contains the following fields, none of which can be directly changed:

General Results

Solution Start Time, Solution End Time

The starting time and ending time of the OPF solution algorithm.

Total Solution Time, Last Solution Status

Time and status of the last OPF solution.

Number of LP Iterations

Total number of LP iterations used during the last OPF solution. The maximum number of iterations is specified in the [Maximum Number of Iterations](#) field of the General Options page.

Initial Cost Function Value

Initial value of the [OPF cost function](#). During the OPF the solution algorithm seeks to minimize the cost function, subject to the equality and inequality constraints.

Final Cost Function Value

Final value of the OPF cost function.

Final Slack Cost Value

The slack cost value is an artificial cost that is only non-zero when there are one or more unenforceable constraints.

Final Total Cost Value

The addition of the final cost function value and the final slack cost value.

Number of Buses in OPF

This field contains the total number of buses that are in areas or super areas that are on OPF control. Thus this field need not be equal to the total number of buses in the case. Marginal costs are only calculated for buses in OPF controlled areas or super areas.

Highest Bus Marginal Cost, Lowest Bus Marginal Cost, Average Bus Marginal Cost

Highest, lowest and average marginal cost for the buses that are in OPF controlled areas or super areas.

Bus MC Standard Deviation

The standard deviation of the Bus Marginal Cost.

Area and Superarea Constraints

Unenforceable Area Constraints

Number of unenforceable area constraints.

Unenforceable Superarea Constraints

Number of unenforceable superarea constraints.

Line MVA Constraints

The **Line MVA Constraints** fields present results associated with the enforcement of the line MVA constraints.

Number of Initial Violations, MVA Sum of Initial Violations

Total number of lines that initially exceeded their MVA limits and were eligible for enforcement by the OPF. For these lines only, the **MVA Sum of Initial Violations** field contains sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

Number of Binding Lines

Total number of lines that are constrained to their limit value.

Highest Line MVA Marginal Cost

The highest Marginal Cost for an MVA change on a line.

Number of Unenforceable Violations

Total number of lines whose MVA flows can not be enforced by the OPF using the available controls.

MVA Sum of Unenforceable Violations

For all the unenforceable lines, this field contains the sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

Interface MW Constraints

The **Interface MW Constraints** field present results associated with the enforcement of the interface MW constraints.

Number of Initial Violations, MW Sum of Initial Violations

Total number of interfaces that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these interfaces only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

Number of Binding Interfaces

Total number of interfaces that are constrained to their limit value.

Highest Interface MW Marginal Cost

The highest Marginal Cost for an MVA change on an interface.

Number of Unenforceable Violations

Total number of interfaces whose MW flows can not be enforced by the OPF using the available controls.

MW Sum of Unenforceable Violations

For all the unenforceable interfaces, this field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

Generator MW Control Limit Violations

The **Generator MW Control Limit Violations** fields present results associated with the enforcement of the generator MW limit constraints.

Number of Initial Violations, MW Sum of Initial Violations

Total number of generators that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these generators only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the generators' actual MW output minus the generators' MW limits.

Number of Unenforceable Violations

Total number of generators whose MW limits can not be enforced by the OPF using the available controls.

MW Sum of Unenforceable Violations

For all the generators with unenforceable limits, this field contains the sum of the absolute values of the generators' actual MW output minus the generators MW limits.

Transformer Regulation Constraints

The **Transformer Regulation Constraints** fields present results associated with the enforcement of phase shifting transformer regulation.

Number of Initial Violations

Total number of phase shifting transformers that initially exceeded their MW set point limits and were eligible for enforcement by the OPF.

Number of Binding Constraints

Total number of phase shifting transformers that are set at their minimum or maximum regulation value.

Number of Unenforceable Violations

Total number of phase shifting transformers whose MW regulation limits can not be enforced by the OPF using the available controls.

Fast Start Generators

The **Fast Start Generators** fields present results associated with the commitment/decommitment of fast start generators.

Number of Generators Turned On

Total number of fast start generators turned on during the OPF solution

Number of Generators Turned Off

Total number of fast start generators turned off during the OPF solution

OPF Options: All LP Variables

The OPF Dialog, All LP Basic Variables page displays the basic and non-basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables, showing the following fields for each:

ID

Variable identifier.

Original Value

The initial value of the LP variable before OPF optimization.

Value

The final value of the LP variable after OPF optimization.

Delta Value

The difference between the original value field and the value field.

BasicVar

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic.

NonBasicVar

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

Cost(Down)

The cost associated with decreasing the LP variable. The field will also show if the variable is at its max or min limit.

Cost(Up)

The cost associated with increasing the LP variable. The field will also show if the variable is at its max or min limit.

Down Range

The available range to decrease the basic variable before a new constraint is hit.

Up Range

The available range to increase the basic variable before a new constraint is hit.

Reduced Cost Up

The cost reduction that would be experienced if an LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

Reduced Cost Down

The cost reduction that would be experienced if an LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

At Breakpoint?

Yes, if the LP variable is at a break point.

OPF Options: LP Basic Variables

The OPF Dialog, LP Basic Variables page displays the basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the basic variables, showing the following fields for each:

ID

Basic variable identifier.

Original Value

The initial value of the basic LP variable before the OPF optimization.

Value

The final value of the basic LP variable after the OPF optimization.

Delta Value

The difference between the original value field and the value of the basic variable.

Basic Var

Shows the indices of the basic variables in the LP basis.

Cost(Up)

The cost associated with increasing the basic variable.

Down Range

The available range to decrease the basic variable before a new constraint is hit.

Up Range

The available range to increase the basic variable before a new constraint is hit.

OPF Options: LP Basis Matrix

The OPF Dialog, LP Basis Matrix page displays the basis matrix associated with the final LP solution. This page is usually only of interest to users interested in knowing the specifics of the LP solution. Knowing the basis matrix can be helpful in figuring out why a particular power system is exhibiting a particular behavior. The rows of the basis matrix are the binding constraints, while the columns of the basis matrix are the basic variables. The entries in the basis matrix then give the sensitivity of each constraint to each of the basic variables. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, you can right-click anywhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

OPF Options: Bus MW Marginal Price Details

The Bus MW Marginal Price Details page displays a grid containing the MW marginal prices computed for an OPF solution. If no OPF solution has been run, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).

OPF Options: Bus MVAR Marginal Price Details

The Bus MVAR Marginal Price Details page displays a grid containing the MVAR marginal prices computed for an OPF solution. If no OPF solution has been run, or if the option to compute MVAR marginal prices has not been selected, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

OPF Options: Bus Marginal Controls

This display shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

OPF Options: Inverse of LP Basis

The OPF Dialog, LP Basis Matrix page displays the inverse of the basis matrix. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, right-click somewhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

OPF Options: Trace Solution

The OPF Dialog, Trace Solution page will trace the LP OPF of the Solution of either the basic variables values or all the store variable values. This page is usually only of interest to users interested in the specifics of the LP solution during the OPF run. By selecting *Do not trace solution*, no LP OPF solution will be tracked when performing an OPF. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the basic variables, showing the following fields for each:

ID

Basic variable identifier.

OPF Area Records

Displays OPF specific information about each area record in the case. To show this display select **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) or **OPF Areas** from the **OPF Case Info** menu found in the **Optimal Power Flow (OPF)** ribbon group on the [Add Ons ribbon tab](#) while in Run mode. The OPF Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display. By default the display contains the following fields:

Number, Name

Area's integer number and its alphanumeric identifier, 255 characters maximum.

AGC Status

Area's automatic generation control status. This is the same field shown on the [Area Records Display](#). The field indicates whether or not the area's generation is changing automatically to control the area interchange.

To be included in the OPF, this field **MUST** be *OPF*. The generation costs for areas that are on *OPF* control are included in the [OPF objective function](#); otherwise they are not. Note that if the area is part of a super area that is on AGC control, this field value is ignored.

Double-click on the field to toggle its value.

XF Phase

Specifies whether phase shifting transformers in the area are available as controls. If *Yes* then all transformers in the area which have their Automatic Control Active are available for control; the Automatic Control status for a transformer is set on the [Branch Information Dialog](#). If *No* then no transformers in the area are available for control.

Branch MVA

Specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included, Line/Transformer MVA Limit Enforcement must not be disabled on the [OPF Options and Results Dialog](#), and the individual line/transformer must be enabled for enforcement on the [OPF Line/Transformer display](#).

Interface MW

Specifies whether or not the MW limits should be enforced for interfaces that have at least one element in this area. For an interface to be included, Interface MW Limit Enforcement must not be disabled on the [OPF Options and Results Dialog](#), and the individual interfaces must be enabled for enforcement on the [OPF Interface display](#).

Load MW Dispatch

Specifies whether or not the MW load demand in an area should be included as available for re-dispatch during an OPF solution. In order for loads to be included in OPF re-dispatch, each individual load within the area must be available for control, and have either a fixed cost benefit or a piecewise-linear cost benefit curve provided. The option to Disable All Load MW Controls must also not be selected on the [OPF Options and Results Dialog](#).

DC Line MW Control

Specifies whether or not DC line MW setpoints can be modified for DC lines that have at least one terminal in this area. For a DC line to be included, DC Transmission Line MW Controls must not be disabled on the [OPF Options and Results Dialog](#).

Include Marg. Losses

Specifies whether or not marginal losses should be included for the area during the OPF solution.

MW Marg. Cost Ave

For an OPF solved case this field shows the average of the bus MW marginal costs for all the buses in the area. If there is no congestion then all of the marginal costs should be equal.

MW Marg. Cost St.Dev., Min., Max.

For an OPF solved case these fields show the standard deviation of the bus MW marginal costs for all the buses in the area, the minimum and the maximum bus MW marginal costs.

Report Limits

Specifies whether or not the kV limits should be reported. If this is set to *NO*, all of the other area options described above will be ignored because the area will not be included in any monitoring during the OPF.

Report Min kV, Report Max kV

Specifies the values for minimum and maximum kV levels to report. Defaults are 0 and 9999. This allows further determination of what elements that belong to the area will be included in the OPF enforcement. Only elements with nominal voltages in the specified Min/Max kV range will be included.

OPF Bus Records

Displays OPF specific information about each bus record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#). The OPF Bus Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields

Number, Name

Bus's number, between 1 and 2,147,483,647 (equals 2^{31} minus 1), and its alphanumeric identifier.

Area Name

Name of the bus's area.

MW Marg. Cost

Marginal change in the objective function for a one MW change in the real power load at the bus.

MVR Marg. Cost

Marginal change in the objective function for a one Mvar change in the reactive load at the bus.

Volt Marg. Cost

Marginal change in the objective function for a 0.01 per unit change in the voltage setpoint for the bus. This field is only valid at bus's whose terminal voltage is controlled by one or more generators.

Note from the developers – this field is still under construction – do not use it yet.

OPF Generator Records

Displays OPF specific information about each generator record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Generator Records** in the [Model Explorer](#). The OPF Generator Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

ID

Alphanumeric ID used to distinguish multiple generators at the same bus.

Area Name of Gen

Name of the generator's area.

AGC

Designates whether the generator's real power output is governed by automatic generation control (AGC). If the AGC field is set to Yes the generator is on AGC in the standard power flow. When a generator is on AGC its real power output is varied automatically, provided the generator is part of an area or super area that is also on automatic control.

In Simulator OPF the default operating mode is that only generators on AGC control are eligible to be OPF controls. In addition, the generator's area or super area must have **AGC Status** of "OPF". However in rare instances you may wish to always make a generator available for control or never make the generator available for control. This value is specified using **OPF MW Control** field.

Fast Start

Designates whether the generator is available as a Fast Start generator during the OPF solution process. Fast start generators are another type of control available to the Optimal Power Flow solution routine. The OPF routine can determine if a generator labeled as a fast start generator would be beneficial in reducing the overall system costs of generation dispatch. If a fast start generator is off-line, but could reduce the cost of the system, then the OPF routine will turn on the generator, and increase the generator's dispatch towards optimizing the system generating cost. Conversely, if a fast start generator is on-line, and the OPF routine determines that reducing the generator's output to 0 would reduce the total generation cost, then the OPF routine will shut off the generator.

Generally speaking, the fast start options should only be used with units with zero Minimum MW limits. Hence it is really aimed at hydro units, or small units which do not need a non-zero minimum MW output for valid operation. This requirement is needed because changing a unit's status is only valid in the OPF routine if it is determined that the unit should dispatch 0 MW to optimize the generating costs of the system.

OPF MW Control

Designates whether the generator's real power output should be included as a control variable in the OPF. This field, which can be toggled, has three possible values:

- "If AGCable" - Generator's control availability depends upon its AGC status.
- "Yes" - Generator is available as a control, regardless of its AGC status.

- "No" - Generator is NOT available as a control, regardless of its AGC status.

Note: in order to be a control the generator must also be in an area or super area on "OPF" control.

Gen MW

The real power output of the generator.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Cost \$/Hr

The total cost of the generator, including the impact of the cost shift and cost multiplier.

MW Marg. Cost

Tells the marginal cost, in \$ / MWhr, to supply one additional MW of load at this bus. If a generator is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the generator's current marginal cost. However the usual case is for the generator to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field values IS NOT equal to the generator's marginal cost.

IC for OPF

Incremental cost of the generator at its current operating point.

Initial MW

The initial real power output of the generator at the beginning of the OPF solution. You can reset the case back to these values by selecting the LP OPF, Restore Previous Control Settings menu item. This menu item is only available following a successful OPF solution.

Initial Cost

The initial generator cost at the beginning of the OPF solution.

Delta MW

Change in the generator's real power output as a result of the OPF.

Delta Cost

Change in the generator's cost as a result of the OPF.

Min MW, Max MW

Minimum and maximum real power output of the generator.

Cost Model

The current cost model being used for the generator. The field value is either "Cubic", indicating that the generator's operating costs are being modeled using a cubic cost function, or "Piecewise Linear", indicating the operating costs are being modeled using a piecewise linear cost function. Toggle the field to change the model. Note that a generator may simultaneously have a cubic model and a piecewise linear model.

Because the OPF uses a linear programming approach, the generator's operating costs are ALWAYS modeled using the piecewise linear model. Generators with an existing cubic cost model are either 1) ignored as OPF controls, or 2) have a piecewise linear cost model automatically created from the cubic model, depending upon the values specified on the [OPF Options and Results Dialog](#).

Cost Curve Points

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero; the generator's costs are being modeled using the cubic function. For such generators you can automatically setup a piecewise linear model simply by entering a non-zero value for the number of points. A piecewise linear model is created that matches as closely as possible the existing cubic model.

Fuel Type

Specifies the fuel type of the generator, if it is known; double-click to toggle through the options. Options are Unknown, Coal, Gas, Hydro, Hydro Pumped, Nuclear, Petroleum, Solar, Wind, and Other.

Profit \$/hr

Shows the profit of the generator. Profit is calculated using this equation:

$$\text{Profit} = (\text{GenMW} * \text{MW Marg Cost}) - [\text{Evaluation of the Generator Cost Function}]$$

OPF Interface Records

Displays OPF specific information about the interface records in the case. To show select **Optimal Power Flow > Results > Interfaces** from the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the Interface records click on the Interfaces tab. This displays the interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

Name

Name of the interface.

Monitor

Specifies whether or not the interface MW limit is enforced in the OPF solution.

Interface MW

The amount of MW flow on the interface.

MW Limit

The interface MW limit.

Percent

The amount of MW flow on the interface as a percentage of the limit.

Monitor Direction

The direction on the interface in which the flow is being monitored.

Monitor Both Directions

If set to **YES**, then the interface flow will be monitored in both directions. Otherwise, it will only be monitored in the **Monitor Direction**.

MW Marg. Cost \$ / MWh

The marginal cost of enforcing the limit on the interface.

Constraint Status

This field will be set to Binding if the interface limit is a binding constraint in the OPF solution, or Unenforceable if the interface limit is unenforceable with the available control set.

OPF Nomogram Records

This list displays OPF specific information about the nomogram records in the case. To show the list, select **Optimal Power Flow > Results > Nomograms** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the nomogram records click on the Nomogram Interfaces tab. This displays the nomogram interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

Nomo. Name

Name of the nomogram.

Nomo. Seg.

The segment of the nomogram. Each segment of the nomogram is treated as a separate constraint in the OPF, and therefore is listed individually in the OPF nomogram table.

Monitor

This field will be set to YES if the nomogram segment is monitored during the OPF.

Interface MW Flow

The total MW flow for the segment of the nomogram.

MW Limit

The segment MW limit.

Percent

The amount of MW flow on the segment as a percentage of the segment's limit.

Monitor Direction

The direction on the segment in which the flow is being monitored.

Monitor Both Directions

Yes or No field indicating if both directions should be monitored at the same time.

MW Marg. Cost \$ / MWh

The marginal cost of enforcing the limit on the segment.

Constraint

This field will be set to Binding if the nomogram limit is a binding constraint in the OPF solution, or Unenforceable if the nomogram limit is unenforceable with the available control set.

OPF Line/Transformer Records

Displays OPF specific information about the line and transformer records in the case. To show the display select **Optimal Power Flow > Results > Branches** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the Line and Transformer records click on the Line/Transformers tab. This displays the Line/Transformer page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

From Number, From Name, From Area Name

Number, Name, and Area Name of the From bus.

To Number, To Name, To Area Name

Number, Name, and Area Name of the To bus.

Circuit

Circuit identifier for the branch.

Monitor

Specifies whether or not the branch's MVA limit will be enforced in the OPF solution.

Max MVA

The maximum MVA flow on the branch. Value is determined based on the end of the branch with the higher MVA flow.

% of MVA Limit (Max)

The maximum MVA flow as a percentage of the branch MVA limit. Value is determined based on the end of the branch with the higher MVA flow.

Lim MVA

The MVA limit for the branch.

MVA Marg. Cost (\$/MVAhr)

The marginal cost of changing the MVA flow on the branch (i.e., if the line rating were to increase by 1 MVA, what savings could be had).

Constraint

This field will be set to Binding if the branch limit is a binding constraint in the OPF solution, or Unenforceable if the branch limit is unenforceable with the available control set.

OPF DC Lines Records

Displays OPF specific information about the DC lines in the case. To show select **Optimal Power Flow > Results > DC Lines** in the [Model Explorer](#). This display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the same fields as the normal [DC Line Records](#) case information display, with the following additional fields:

OPF Control

Set to **YES** if the DC line is to be controlled during the OPF. To be controlled, at least one of the terminals of the DC line must lie in an area that is on OPF control and this must be set to **YES**.

Min MW or amps, Max MW or amps

The minimum and maximum power (or current) setpoints allowed for the DC line.

OPF Load Records

Displays OPF specific information about each load record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Loads** in the [Model Explorer](#). The OPF Load Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Number, Name

Number and name of the bus to which the load is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

ID

Alphanumeric ID used to distinguish multiple loads at the same bus.

Area Name of Load

Name of the load's area.

AGC

Designates whether the load's real power demand is governed by automatic generation control (AGC). If the AGC field is set to Yes the load is on AGC in the standard power flow. When a load is on AGC its real power output is varied automatically during an OPF or SCOPF solution ONLY, provided the load is part of an area or super area that is also on OPF control. Loads will not be dispatched if the area is on some other form of AGC control.

In Simulator OPF the default operating mode is that only loads on AGC control are eligible to be OPF controls. In addition, the load's area or super area must have **AGC Status** of "OPF", and the area must have its Load MW Dispatch set to YES.

MW

The real power output of the load.

Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

Hourly Benefit

The total benefit of the load, including the impact of the cost shift and cost multiplier.

MW Marg. Cost

Tells the marginal cost, in \$ / MWhr, to reduce one additional MW of load at this bus. If a load is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the load's current marginal cost. However the usual case is for the load to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field value IS NOT equal to the load's marginal cost.

Inc. Benefit

Incremental benefit of the load at it's current operating point.

Initial MW

The initial real power demand of the load at the beginning of the OPF solution.

Initial Cost

The initial load cost at the beginning of the OPF solution.

Delta MW

Change in the load's real power output as a result of the OPF.

Delta Cost

Change in the load's cost as a result of the OPF.

Min MW, Max MW

Minimum and maximum real power demand of the load.

of Benefit Curve Points

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero.

Profit \$/hr

Shows the profit of the load.

OPF Super Area Records

Displays OPF specific information about each super area record in the case. To show this display select **Optimal Power Flow > Results > Super Areas** in the [Model Explorer](#). The OPF Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

Super Area

Alpha-numeric identifier of the super area.

AGC Status

Super area's automatic generation control status. This is the same field shown on the Super Area Records display. The field indicates whether or not the super area's generation is changing automatically to control the super area's interchange.

The super area AGC Status field always overrides the AGC Status for the individual areas, except when it is set to "Off AGC".

To be included in the OPF this field MUST be "OPF". In that case the generation costs for all the areas in the super area are included in the OPF objective function. Otherwise they are only included if the super area AGC Status is "Off AGC" and their particular area's AGC Status is on "OPF". Double-click on the field to toggle its value.

Num Areas

Number of areas in the super area. To see the individual areas use the local menu to view the Super Area dialog.

Include Marg. Losses

Specifies whether or not to include marginal losses in OPF calculations for this super area.

MW Marg. Cost Ave.

If the super area is on "OPF" control then for a solved case this field shows the average of the bus MW marginal costs for all the buses in the super area. If there is no congestion then all of the marginal costs should be equal.

MW Marg. Cost St. Dev.

Standard deviation of the bus MW marginal costs for the buses in the super area.

ACE MW

Area control error for the super area.

Gen MW, Load MW

Total real power generation and load in the super area.

Total Sched MW, Int MW

Scheduled and actual interchange real power interchange between the super area and the rest of the system. Both of these fields are the algebraic summation of the scheduled and actual interchange for the areas in the super area.

Loss MW

Total real power losses for the super area.

MW Marg. Cost Min, MW Marg. Cost Max

Minimum and maximum of all the bus MW marginal costs for the buses in the super area.

OPF Controls

The following classes of controls are available during the OPF solution. Note, individual classes of controls can be enabled/disabled for the entire case using the OPF Options and Results dialog and for particular areas using the OPF Area Records display. Also, all classes of controls have associated minimum/maximum limits which are always enforced.

Generator MW output and Fast Start commitment

The generator MW outputs are the major control for controlling the MW flow in the network and for minimizing the objective function. Only generators in areas or super areas that are on "OPF" control are eligible for control; otherwise the generator's MW output remains fixed at its initial value. Whether a particular generator is available for control also depends upon the status of its AGC and OPF MW Control fields. These fields are set on the [OPF Generator Records](#) display. For Fast Start Generators, it is also possible to use the startup and shutdown of these generators as a control. See [OPF Options: Control Options](#) for more details.

Phase shifting Transformer tap position

Phase shifting transformers are used primarily to control the flow of real power in the network. When phase shifting transformers are controlled in the OPF routine, the phase angle is allowed to move anywhere within the phase angle range of the device in order to help alleviate violations on other branches in the system. The flow on the phase shifter is allowed to violate the prescribed MW range given for the phase shifter, but is NOT allowed to violate the MVA rating of the device.

Additionally, in order for a phase shifter to be included in the OPF solution dispatch, the XF Phase property of the area must be set to YES. Each individual phase shifter also has a property for being included in the OPF control that must be turned on in order for the phase shifter to participate in the OPF dispatch. This option can be set for a phase shifting transformer by opening its [Information Dialog](#) and checking the OPF Phase Shifter Control options on the OPF page of the dialog.

Load MW Dispatch

The load MW demands can also be included as controls for re-dispatch during an OPF solution. The concept of controlling a load is generally the same as controlling a generator. Loads can be assigned piecewise linear benefit curves, and included in the OPF dispatch algorithm. Only areas whose AGC control is set to "OPF" are eligible for control. Furthermore, the OPF area's Load MW Dispatch property must also be set to YES. Each area load that is to be included for OPF dispatch must have a benefit model defined, and have its Available for AGC field set to YES. These fields can be set in the [OPF Load Records](#) display.

DC Line MW setpoint

DC Line MW setpoints can be used as controls in the OPF. See [OPF DC Lines Records](#).

OPF Phase Shifter Records

Displays OPF specific information about each phase shifter record in the case. To show this display, select **Optimal Power Flow > Results > Phase Shifters** in the [Model Explorer](#). The OPF Phase Shifter Records display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields:

From Number, From Name

The name and number of the bus at the From end of the phase shifter.

To Number, To Name

The name and number of the bus at the To end of the phase shifter.

Circuit

The circuit identifier for the phase shifter.

OPF Control

Specifies whether or not the phase shifter is available for control during an OPF solution.

Area PS Control

Specifies if automatic phase shifter control has been enabled for the area containing the particular phase shifting transformer. If the area phase shifter control is disabled, all phase shifting transformers within the area will remain fixed at their initial settings during the entirety of the OPF solution process. This setting overrides the individual automatic control settings of each phase shifting transformer within the area.

XF Auto

Specifies if the transformer automatic control is enabled. If an individual transformer's automatic control is disabled, it will remain at its initial settings during the entirety of the OPF solution process.

Phase (Deg)

The actual phase shift of the phase shifter, in degrees.

Initial Degrees

The initial phase angle before the OPF solution was calculated.

Delta Degrees

The change in the phase angle during the OPF solution.

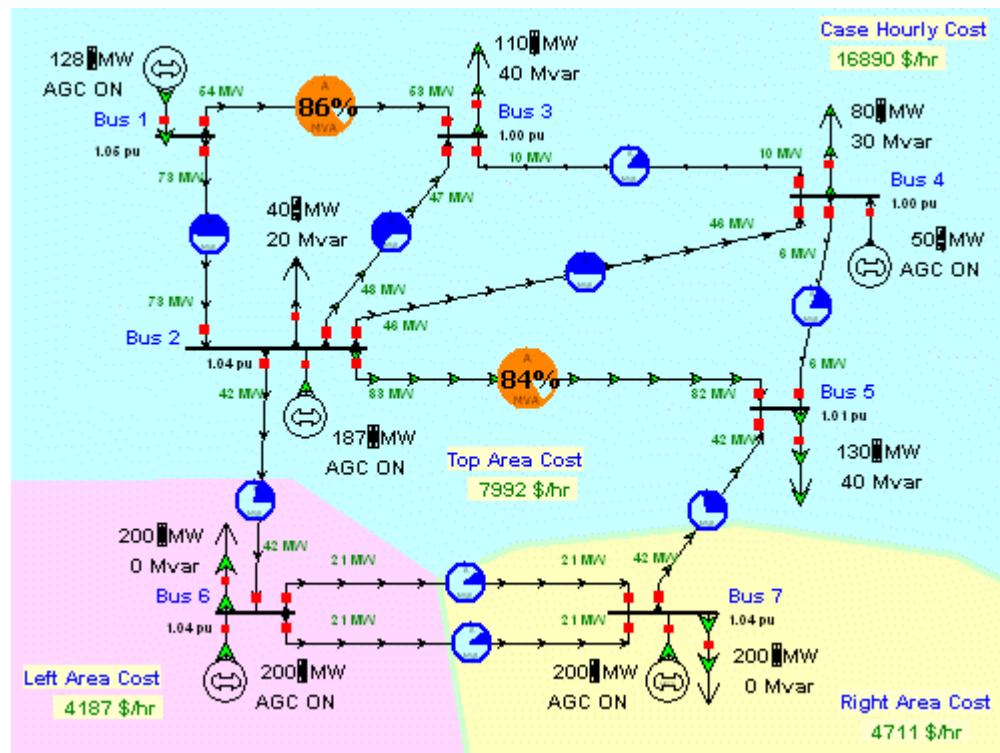
Tap Min, Tap Max

The minimum and maximum tap positions allowed for the phase shifter operation.

OPF Example - Introduction

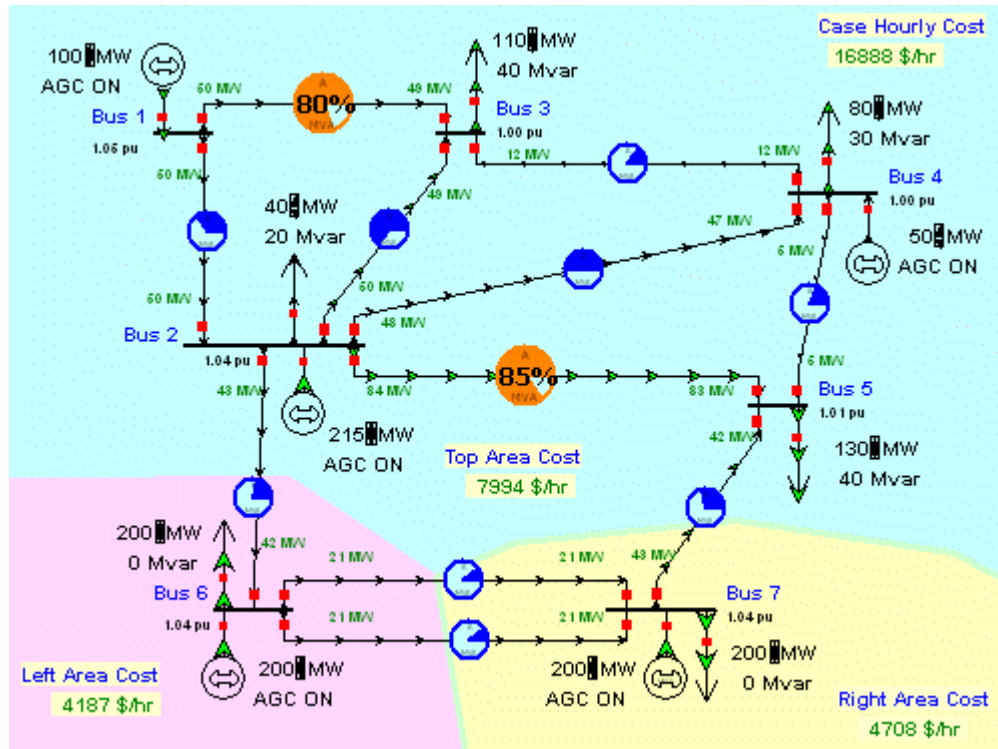
As a simple example of using the OPF, consider the seven bus, three area system contained in the file B7OPF.pwb (included with the PowerWorld Simulator). For this case all three areas are initially on Economic Dispatch (ED) AGC control and hence by default would not be included in the OPF solution. Also, the initial interchange between the areas is equal to zero and the generators are modeled using cubic cost functions.

To initially solve the case using the standard power flow, select **Single Solution** in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. The case should look like the following figure:



B7OPF Case Solved using Economic Dispatch

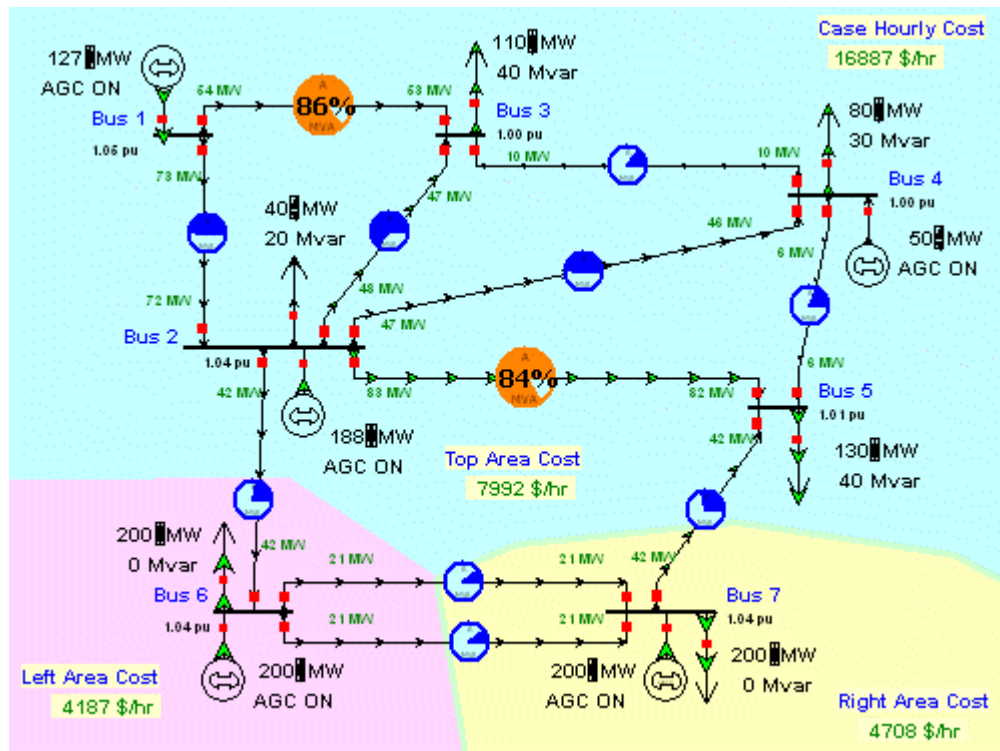
Now we'll modify the case to set the three areas for OPF control. To do this, select **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) to display the [OPF Area Records display](#). Toggle the AGC status for each of the three areas to change it to "OPF". Now select **Primal LP** in the **OPF** ribbon group on the [Add Ons](#) ribbon tab to solve the case using the LP OPF. The results should look similar to the figure below. The one-line shows the hourly cost for each area and the total case hourly cost, equal to \$ 16,888 / hr.



B7OPF Case Solved using LP OPF

Note that the results are very similar but not identical to the economic dispatch case. We would expect the cases to be similar since for cases with no congestion the OPF solution should be (ideally) equal to the economic dispatch solution. The difference between the two is because in the LP OPF the generator cost functions are converted from a cubic model to a piece-wise linear model using a user specified number of segments, which is 5 segments by default. This value can be viewed/modified from the Control Options page of the [OPF Options and Results display](#).

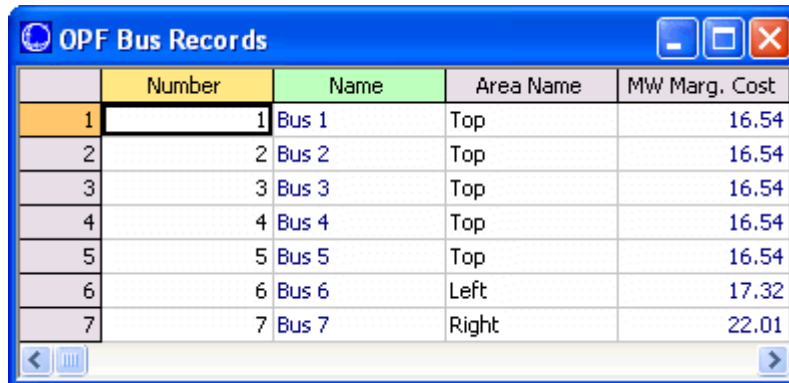
Change the **Total Points Per Cost Curve** field to 100 and resolve. The results are shown in the figure below, which now are almost identical to the economic dispatch results. The disadvantage in using a large number of cost segments is that it degrades the performance of the LP OPF slightly on larger cases.



B7OPF Case Solved using LP OPF with 100 Cost Segments for Generators

OPF Example - Marginal Costs

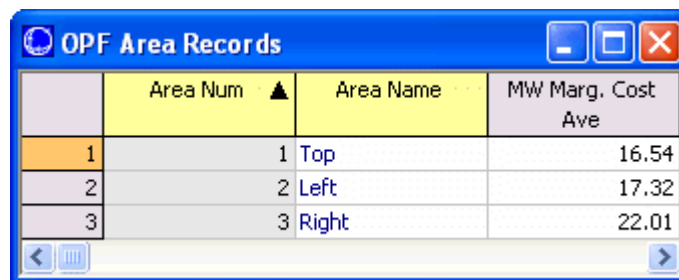
Using the OPF solution from the previous page, select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#) to view the bus marginal costs and **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) to view the area marginal costs. The results should be as shown below.



The screenshot shows a window titled "OPF Bus Records" with a table containing 7 rows of bus data. The columns are Number, Name, Area Name, and MW Marg. Cost. The first row is highlighted in orange.

	Number	Name	Area Name	MW Marg. Cost
1	1	Bus 1	Top	16.54
2	2	Bus 2	Top	16.54
3	3	Bus 3	Top	16.54
4	4	Bus 4	Top	16.54
5	5	Bus 5	Top	16.54
6	6	Bus 6	Left	17.32
7	7	Bus 7	Right	22.01

Seven Bus Case Bus Marginal Costs



The screenshot shows a window titled "OPF Area Records" with a table containing 3 rows of area data. The columns are Area Num, Area Name, and MW Marg. Cost Ave. The first row is highlighted in orange.

	Area Num	Area Name	MW Marg. Cost Ave
1	1	Top	16.54
2	2	Left	17.32
3	3	Right	22.01

Seven Bus Case Area Marginal Costs

Note that the marginal costs for all the buses in an area are identical to the area's MW marginal cost. This is the expected result for systems without any line congestion. The area MW marginal costs are not identical. This is because currently each area is independently enforcing its own MW interchange. In the next example we'll jointly dispatch the three areas by combining them into a single [super area](#).

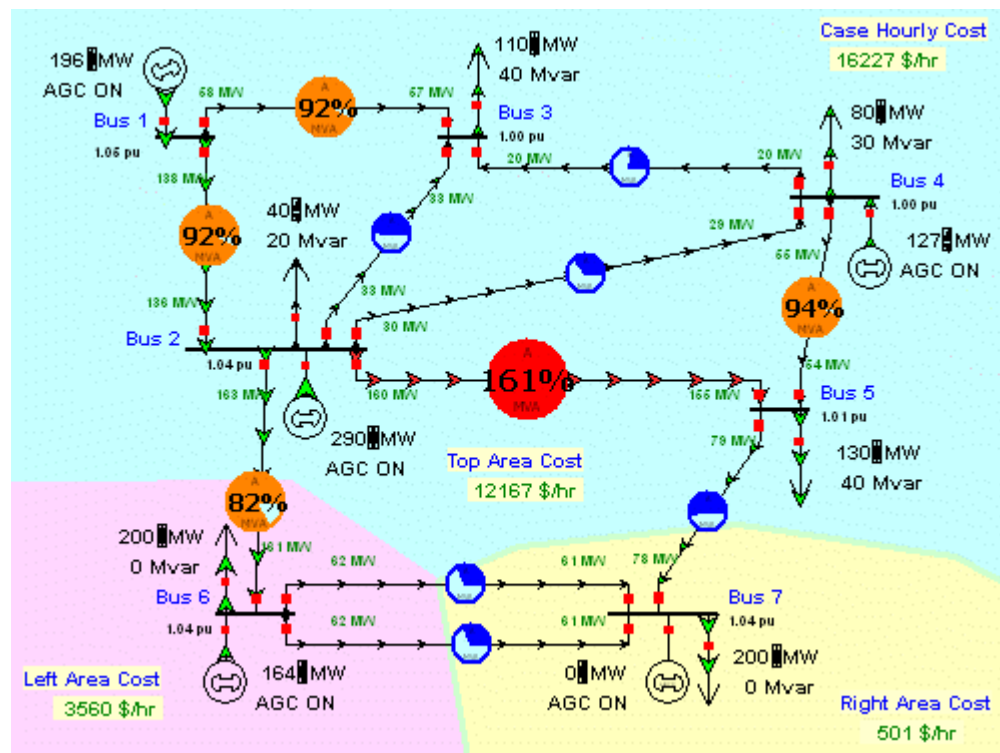
OPF Example - Super Areas

To jointly dispatch the three areas we'll first combine them into a single super area. To setup the super area first select **Optimal Power Flow > Results > Super Areas** from the [Model Explorer](#) to display [the OPF Super Area Records display](#) (alternatively you could also use the **Aggregations > Super Areas** display). To enter a new super area, right click on the "None Defined" entry in the first row of the display to show the display's local menu. Select **Insert**. This displays the [super area dialog](#). Select **Rename** and enter a name for the new super area, "ThreeAreas." To add the three areas to the new super area enter "1-3" in the **New Area #'s** field and then select **Add New Areas by Number** (alternatively you could select the areas from the **New Area Name** list). Also, to enable the super area for control, set the **Super Area Control Options** field (AGC Status) to Optimal Power Flow Control. Select the **Ok** button to save the new super area.

Before resolving the OPF, let's temporarily disable enforcement of line MVA constraints. You can do this from the [Constraint Options page](#) of the [OPF Options and Results dialog](#). Check the **Disable Line/Transformer MVA Limit Enforcement**.

Also, now would be a good time to save the changes. To avoid overwriting the existing B7OPF file, select **Save Case As** from the [Application button](#) to save the case (pwb and pwd files) with a different name, say B7OPFSA (SA for super area).

Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to resolve the OPF. The results should be as shown below.

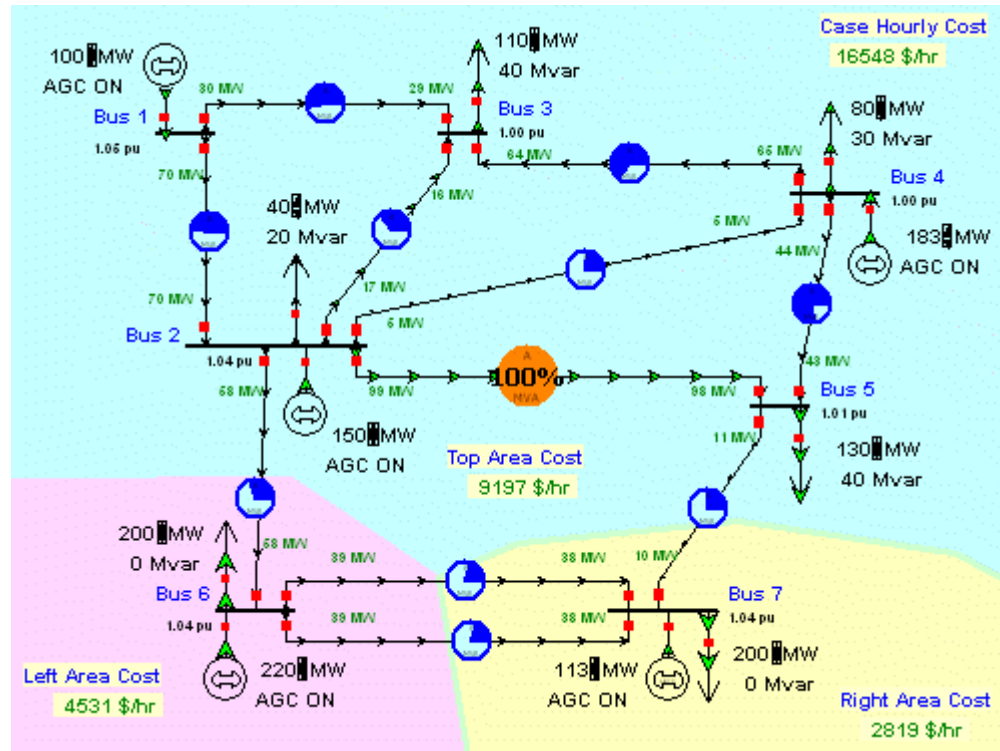


system using the super area is that now there are line violations. These violations will be removed next by enforcing the line MVA constraints.

OPF Example - Enforcing Line MVA Constraints

To remove the line MVA violations go back to the [Constraint Options Page](#) of the [OPF Options and Results](#) dialog. Uncheck the **Disable Line/Transformer MVA Limit Enforcement**.

Now resolve the LP OPF, enforcing the line constraints. The resultant solution is shown below.



OPF Solution with Super Area WITH Line MVA Constraint Enforcement

With line constraint enforcement active the OPF optimally redispatches the generation taking into account the line MVA limits. However, enforcing these line constraints comes at a cost. Notice that the total case hourly cost has increased from \$ 16,227 / hr to \$ 16,548 / hr, which is still substantially less than the \$ 16,887 / hr figure we had for the case without the superarea.

Enforcing the line constraints also has an impact on the bus marginal costs, shown below.

OPF Bus Records				
	Number	Name	Area Name	MW Marg. Cost
1	1	Bus 1	Top	15.97
2	2	Bus 2	Top	15.63
3	3	Bus 3	Top	17.00
4	4	Bus 4	Top	17.42
5	5	Bus 5	Top	23.07
6	6	Bus 6	Left	17.44
7	7	Bus 7	Right	21.11

Impact of Line MVA Enforcement on the Bus Marginal Costs

The actual marginal cost of enforcing the line constraint can also be viewed on the [OPF Line/Transformer MVA Records](#) display. The MVA Marg. Cost tells the marginal cost of enforcing the

constraint, expressed in units of \$ / MVA / hr. Left-click on the **MVA Marg Cost** field header to sort the display using this field.

OPF Constraints Records

Line/Transformer Constraints

☐ Disable Line/Transformer MVA Limit Enforcement

Percent Correction Tolerance

☐ Only Show Limit Violations

☒ Only Areas with Line MVA Enforcement

☐ Use Area/Zone Filters

☐ Enforce Line/Transformer MW Flow Limits (not MVA)

Interface Constraints

☐ Disable Interface MW Limit Enforcement

Percent Correction Tolerance

☐ Only Show Limit Violations

☐ Use Area/Zone Filters

Monitor/Enforce Contingent Interface Elements

☒ Never

☐ Power Flow/OPF but not CA/SCOPF

☐ All Applications including CA/SCOPF

☐ Do not enforce limits on radials lines and buses

Limit Monitoring Settings ...

If you want to change enforcement percentages, modify the Limit Monitoring Settings

Lines/Transformers | Interfaces

	From Number	From Name	From Area Name	To Number	To Name	To Area Name	Circuit	Monitor	Max MVA	% of MVA Limit (Max)	Lim MVA	MVA Marg Cost ▼
1	2	Bus 2	Top	5	Bus 5	Top	1	YES	100.0	100.0	100.0	13.8
2	6	Bus 6	Left	7	Bus 7	Right	1	YES	39.7	19.9	200.0	
3	1	Bus 1	Top	3	Bus 3	Top	1	YES	34.6	53.3	65.0	
4	3	Bus 3	Top	4	Bus 4	Top	1	YES	64.6	64.6	100.0	
5	2	Bus 2	Top	4	Bus 4	Top	1	YES	23.9	23.9	100.0	
6	7	Bus 7	Right	5	Bus 5	Top	1	YES	50.7	25.4	200.0	
7	2	Bus 2	Top	6	Bus 6	Left	1	YES	59.5	29.7	200.0	
8	6	Bus 6	Left	7	Bus 7	Right	2	YES	39.7	19.9	200.0	
9	1	Bus 1	Top	2	Bus 2	Top	1	YES	70.5	47.0	150.0	
10	4	Bus 4	Top	5	Bus 5	Top	1	YES	45.6	76.0	60.0	
11	2	Bus 2	Top	3	Bus 3	Top	1	YES	28.1	35.1	80.0	

Security Constrained Optimal Power Flow Overview

Note: The SCOPF option in PowerWorld Simulator is only available if you have purchased the SCOPF and OPF add-ons to the base package. To learn more about the SCOPF, please feel free to read through the information contained in these help files. Contact PowerWorld Corporation for details about ordering the SCOPF and OPF versions of Simulator.

The [optimal power flow](#) (OPF) algorithm has the purpose of minimizing an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. Normally however, the secure operation of a power system requires that there be no unmanageable contingency violations. Thus, the minimization of the objective function requires considering contingencies. This is achieved using a security constrained optimal power flow (SCOPF) algorithm.

While the [optimal power flow](#) (OPF) algorithm determines the optimal state of the system by iterating between solving a power flow and solving a linear program that changes the system controls, during the [SCOPF solution process](#) in addition to solving the base case power flow, all contingencies must also be solved. The linear program which then runs changes the system controls to remove contingency violations as well as base case operating violations.

The SCOPF algorithm makes control adjustments to the base case (pre-contingency condition) to prevent violations in the post-contingency conditions. If enough controls are available in the system, the solution minimizes the [objective function](#) and the system enforces contingency violations. If the system does not have enough controls, then some violations may be persistent under certain contingencies. Those represent unenforceable constraints, which result in high bus marginal costs.

The commands and options for the SCOPF are accessed by selecting the **SCOPF** option from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The SCOPF function uses the OPF options defined in the [OPF Options and Results dialog](#) and the contingency settings specified in the [contingency analysis dialog](#). Please read through the OPF and the contingency analysis help for more information about these settings. We encourage you to become familiar with Simulator OPF and the contingency analysis tool before running SCOPF simulations. Other commands in the [SCOPF dialog](#) are used to specify the base case solution process and for accessing the SCOPF results.

SCOPF Objective Function

The SCOPF objective function uses the function defined in the OPF settings. There are two objective functions in Simulator: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute value of the change in the generation in the specified areas or super areas. The objective function is set up in the [OPF Options and Results dialog](#).

The result of the SCOPF will be different from the OPF solution because the SCOPF meets additional [inequality constraints](#) associated with the contingency violations.

SCOPF Dialog

The SCOPF dialog allows the user to control options of Security Constrained OPF, as well as to access the optimization results.

The SCOPF can be run from this dialog using the **Run Full Security Constrained OPF** button at the top of the form.

There are three pages of information included in this form:

[Options](#)

[Results](#)

[LP Solution Details](#)

SCOPF Solution Process

The SCOPF involves three major steps that can be solved either automatically or manually from the SCOPF control dialog:

- Initialization to setup the SCOPF LP tableau and the control structures
- Contingency analysis calculation and storage of control sensitivities associated with each contingency violation
- SCOPF iterations, which include an LP solution and a power flow solution. During each LP step in the LP routine, the algorithm enforces the newest most severe contingency violation. After each violation is processed, all of the unprocessed violations are updated. This step is crucial since often resolving the most severe violation resolves numerous other violations. For instance, a single line might be overloaded in a number of contingencies: fixing the worst contingency fixes the others as well. On the other hand, processing some violations may result in new violations. In order to verify that no new violations have been created by the control changes made, the SCOPF will go back to step two and reprocess all the contingencies and the new base solution. The number of times this iteration occurs is determined by the Maximum Number of Outer Loop Iterations.

The SCOPF terminates when all the contingency violations have been processed. Note that the user can rerun the SCOPF by repeating the solution process if they want to verify the contingency violation enforcement at the new optimal operating point.

SCOPF Results

The Results page of the [Security Constrained Optimal Power Flow](#) form provides information on the results of the latest SCOPF solution, including the [contingency violations](#) included, the [marginal price details](#), and the [marginal control details](#).

SCOPF Equality and Inequality Constraints

Two general types of constraints are involved in the SCOPF solution: equality and inequality constraints. Equality constraints are constraints that have to be enforced. That is, they are always "binding". For example in the SCOPF, as well as in the OPF and in the power flow, the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance). In contrast, inequality constraints may or may not be binding. For example, a line MVA flow under a certain contingency may or may not be at its limit.

The SCOPF problem is solved by iterating between a power flow solution and a contingency constrained LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiates the SCOPF from a standard power flow and from the OPF are the constraints that are explicitly enforced by the LP solver. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)

SCOPF Equality Constraints

The SCOPF equality constraints are the same as the [OPF equality constraints](#): Area MW interchange, bus MW and Mvar power balance, Generator voltage setpoint and super area MW interchange.

SCOPF Inequality Constraints

The following classes of inequality constraints are enforced during the SCOPF solution.

Generator real power limits

Generator real power limits are enforced during the SCOPF LP solution.

Generator reactive power limits

Generator reactive power limits are enforced during the SCOPF LP solution.

Interface MW Limits

Interface MW limits are enforced during the SCOPF solution. [Interfaces](#) are used to represent the aggregate flow through a number of different devices. During the SCOPF the MW post-contingency flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Constraints Dialog](#) or the [OPF interfaces records](#). As default, the interface enforcement is not disabled. Note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from the [OPF Area Records display](#). Note: the default is that interface enforcement is not active, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be specified from the [OPF interfaces records](#) display or in the [Limit Monitoring Settings](#) dialog. The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of contingency enforcement. When manually solving the SCOPF one can skip a contingency violation associated to the interface by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the SCOPF solution. During the LP the post-contingency branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF constraints dialog](#) or the [OPF Line/Transformer Records](#) display. The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. Enforcement for tie-lines must be active for either area. This field can be set from the [OPF Line/Transformer Records](#) display. The default is that branch enforcement is not active, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records](#) display or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding under contingency conditions. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcing the contingency constraint, which is shown on the [SCOPF Bus Marginal Price Details](#) dialog. When manually solving the SCOPF one can skip a contingency violation associated to the branch by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

SCOPF Control

The control dialog allows the user to manually or automatically run a SCOPF simulation and visualize the execution of the simulation. The [SCOPF solution process](#) involves three steps: base case solution and initialization, contingency analysis, and SCOPF iterations.

Run Full Security Constrained OPF

Press this button to run the three steps of the SCOPF solution automatically. The SCOPF will solve the base case using the selected method, will use the currently stored list of contingencies during the contingency analysis step, and will take the CA results and sensitivities to iterate in order to obtain the optimal solution that minimizes cost and enforces contingency violations.

Options

Maximum Number of Outer Loop Iterations

Indicates the number of maximum outer loop iterations. The outer loop iterations determines how many times the contingency analysis will be re-run following a successful SCOPF dispatch. In this manner, Simulator will look for new violations that may occur due to the new generation dispatch.

Consider Binding Contingent Violations from Last SCOPF Solution

When checked, this option ensures that the contingent violations from the last SCOPF solution are included in the current SCOPF solution. This option is helpful in preventing the SCOPF from hunting between having a constraint binding in one solution, and resolving with it not binding in a later solution because it was previously remedied. This option should generally always be checked, unless the user is sure that the previous solution has no bearing on the current solution, such as having made major changes to the system since the previous solution.

Initialize SCOPF with Previously Binding Constraints

When checked, this option results in the SCOPF solution process starts with the exact same LP tableau from the last solution. This can make for fairly fast solutions (recognizing that the contingency analysis needs to be resolved) when the changes to the system are small. Simulator automatically uses this option when doing multiple outer loops of the SCOPF. This option allows the user to solve the outer loops (set the outer loop counter to 1) by repeatedly solving the SCOPF manually, potentially making modifications between solutions if desired.

Set Solution as Contingency Analysis Reference Case

Check this field to set the solution of the SCOPF as the contingency analysis reference. If the system has enough controls to remove all the contingency violations, a rerun of the contingency [analysis](#) using the SCOPF solution as the reference should report no branch violations.

Maximum Number of Contingency Violations Allow Per Element

Specify the maximum number of contingency violations that the SCOPF analysis should allow per element. After completing the power flow solutions for the base case and the contingencies, only this number of contingency violations per element are then passed to the linear programming algorithm. The assumption is that if you fix the 10 worst contingency violations for an element then the others will also be fixed.

Basecase Solution Method

Specifies whether the solution of the base case is performed using the power flow algorithm or the optimal power flow algorithm. The selection will affect the initial conditions of the system and consequently the contingency analysis results and the sensitivities used by the LP solver. Currently

the SCOPF does not resolve the contingency analysis during the optimization since this is computationally expensive. See the [SCOPF solution process](#) for details.

Handling of Contingent Violations Due to Radial Load

It is often common when computing a security constrained OPF to have violations occur on branches due to radial load. In these instances, there is no way to adjust controls to continue to serve the load, without overloading the serving element. Therefore you can choose how contingent violations of this type should be handled by the SCOPF. You can choose to flag them but not include them in the SCOPF, ignore them completely, or include the violations in the SCOPF. Note that if you include the violations in the SCOPF, the SCOPF algorithm will not be able to remove the violation on the element via generation dispatch. However, it may be able to do so if the load in question has a load benefit curve defined and is available for load shed dispatch in the SCOPF routine.

DC SCOPF Options

The DC options given for the SCOPF revolve around the treatment of Line Outage Distribution Factors (LODF) during the DC SCOPF solution. You can choose to discard the LODFs when the SCOPF is finished, store them in memory (lost when Simulator is closed,) or in memory and in the PWB file (if saved.)

If the LODFs have been stored and you wish to clear them (they can require quite a bit of RAM depending on the number of contingencies and size of the case,) you can press the button labeled **Clear Stored Contingency Analysis LODFs**.

SCOPF Results Summary

Number of Outer Loop Iterations

The number of outer loop iterations required to solve the SCOPF.

Number of Contingent Violations

Number of violations from the contingency analysis portion of the solution, which are used to attempt to determine the security constrained dispatch.

SCOPF Start Time, SCOPF End Time

Physical time when the SCOPF solution process started and finished.

Total Solution Time (Seconds)

Length of time needed to determine the SCOPF solution.

Total LP Iterations

Total number of Linear Programming iterations necessary to determine the SCOPF solution.

Contingency Analysis Input

Number of Active Contingencies

The number of contingencies included in the SCOPF simulation. Specific contingencies may be excluded from the simulation in the [contingency analysis dialog](#) by changing the skip field of a contingency to YES. The contingency analysis dialog can be conveniently accessed from the SCOPF control dialog.

View Contingency Analysis Form

Clicking this button shows the [contingency analysis dialog](#).

Contingency Analysis Results

This window allows the user to monitor the details of the contingency analysis step. During contingencies, the outage actions, the solution of each contingency, and the solution of the CA run are reported.

SCOPF CTG Violations

The contingency violations page is available from the Results tab of the [Security Constrained Optimal Power Flow](#) form. The display lists the results from the latest contingency analysis run including the violations that were included in the SCOPF and the final error for each violation. This dialog may change if the user reruns the SCOPF by solving the contingency analysis using the SCOPF solution as contingency analysis reference. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display shows the following fields:

Contingency Name

This is the contingency label. By default single line contingencies start with an "L", single generator outages with a "G", and single transformer outages start with an "X".

Category

Currently, the SCOPF considers only branch and interface violations. Thus, the category of the contingencies should be branch MVA or Interface.

Element

Shows information about the specific element that presented the violation. When the violation occurs in a branch, this column includes the identifiers of the sending and receiving ends of the branch, the circuit, and the direction of the violating flow. Since Interfaces are directed, this field will present only the interface name in the case of violating interfaces.

Value

The percentage flow that appears in the branch during the contingency prior to optimization. If this number is larger than the scaled limit, the violation needs to be removed.

Scaled Limit

The scaled limit corresponds to the Line/Transformer Percentage specified in the [limit monitoring settings](#) dialog. By specifying this limit to be higher than 100% some of the contingency violations might be effectively relaxed. Sometimes this helps the OPF and the SCOPF obtain a feasible solution. On the other hand, it is often required to analyze the performance of the system if branches would have higher ratings.

New Value

The percentage flow that appears in the branch during the contingency after SCOPF optimization. If this number is larger than the scaled limit, the contingency violation has not been removed and it is therefore unenforceable. If the value is equal to the scaled limit, then the contingency violation constraint would be binding. If the value is smaller than the scaled limit, the contingency violation has been removed.

Error

The difference between the new value and the scaled limit. If the error is positive, the line is unenforceable. If the error is zero, the constraint has been corrected.

Included

Indicates if the contingency violation was included as a constraint in the SCOPF solution.

Marginal Cost

Indicates the cost associated with the contingency violation. If the constraint is unenforceable, the marginal cost is assigned arbitrarily as a high value in the [OPF constraint options](#) dialog.

Unenforceable

Indicates whether the contingency violations is unenforceable, i.e., the system has not enough controls to relieve the branch overload when the contingency occurs.

Skip Violation?

Change this field to NO if the contingency violations should not be included as a SCOPF constraint. This is sometimes useful in order to analyze the effect of the contingency violation in the SCOPF solution. This field may be toggled when doing a manual SCOPF solution.

SCOPF LP Solution Details

The LP Solution Details page of the [Security Constrained Optimal Power Flow](#) form provides information on the linear program solution of the SCOPF, including a list of [All LP Variables](#), [LP Basic Variables](#), and [LP Basis Matrix](#). This information applies to the linear programming tableau solution method.

SCOPF All LP Variables

The SCOPF All LP Variables dialog displays the basic and non-basic variables associated with the final LP SCOPF solution. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. To see the display, open the [Security Constrained Optimal Power Flow](#) form, click on the LP Solution Details tab, and access the All LP Variables page.

Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables with the following fields:

ID

Variable identifier.

Org. Value

The initial value of the LP variable before SCOPF optimization.

Value

The final value of the LP variable after SCOPF optimization.

Delta Value

The difference between the original value field and the value field.

Basic Var

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic. These values are set up after the SCOPF calculates the contingency violation sensitivities.

NonBasicVar

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

Cost(Down)

The cost associated with decreasing the LP variable. The field will show if the variable is at its max or min limit.

Cost(Up)

The cost associated with increasing the LP variable. The field will show if the variable is at its max or min limit.

Down Range

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

Up Range

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

Reduced Cost Up

The cost reduction that would be experimented if a LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

Reduced Cost Down

The cost reduction that would be experimented if a LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

At Breakpoint

Yes, if the LP variable is at a break point.

SCOPF LP Basic Variables

The LP Basic Variables (from the SCOPF [LP Solution Details](#) page) displays the basic variables of the final LP solution. The basic variables may correspond to controls that can be altered to minimize the objective function, or slack variables associated with unenforceable constraints. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each LP variable with the following fields:

ID

Basic variable identifier.

Org. Value

The initial value of the basic LP variable before the SCOPF optimization.

Value

The final value of the basic LP variable after the SCOPF optimization.

Delta Value

The difference between the original value field and the value of the basic variable.

Basic Var

Shows the indices of the basic variables in the LP basis.

Cost(Up)

The cost associated with increasing the basic variable.

Down Range

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

Up Range

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

SCOPF LP Basis Matrix

The LP Basis Matrix page of the SCOPF [LP Solution Details](#) tab displays the basis matrix associated with the final SCOPF LP solution. There is one row per constraint and one column per basic variable. Additional columns summarize information associated with each constraint. This page is usually only of interest to users interested in knowing the specifics of the SCOPF solution. Knowing the basis matrix can be helpful in figuring out why a particular SCOPF solution exhibits a certain behavior. The entries in the basis matrix give the sensitivity of each constraint to each of the basic variables.

As any case info display in simulator, right click to see options to copy information to the clipboard and to perform standard windows actions, such as printing.

SCOPF Bus Marginal Price Details

This display (from the SCOPF [Results](#) page) shows information about the components of the marginal cost at each bus. The display is relevant to see how the bus marginal cost depends on the cost of enforcing system constraints such as branch limits and area equality constraints. This display is useful for indicating which constraints are contributing towards the determination of the marginal price at each bus.

As any case info display in Simulator this display can be customized and the information copied, printed, and saved by accessing the local menu option with the mouse right click.

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).

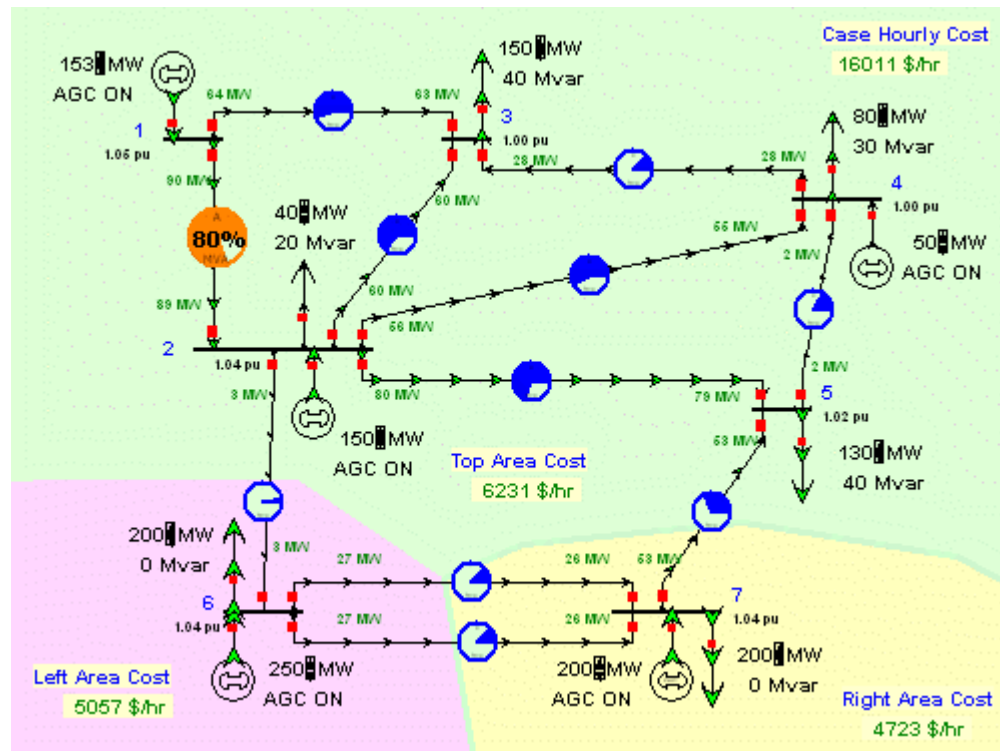
SCOPF Bus Marginal Controls

This display (from the SCOPF [Results](#) page) shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

The Marginal Controls page can be accessed by opening the [Security Constrained Optimal Power Flow](#) form, and clicking on the Results tab.

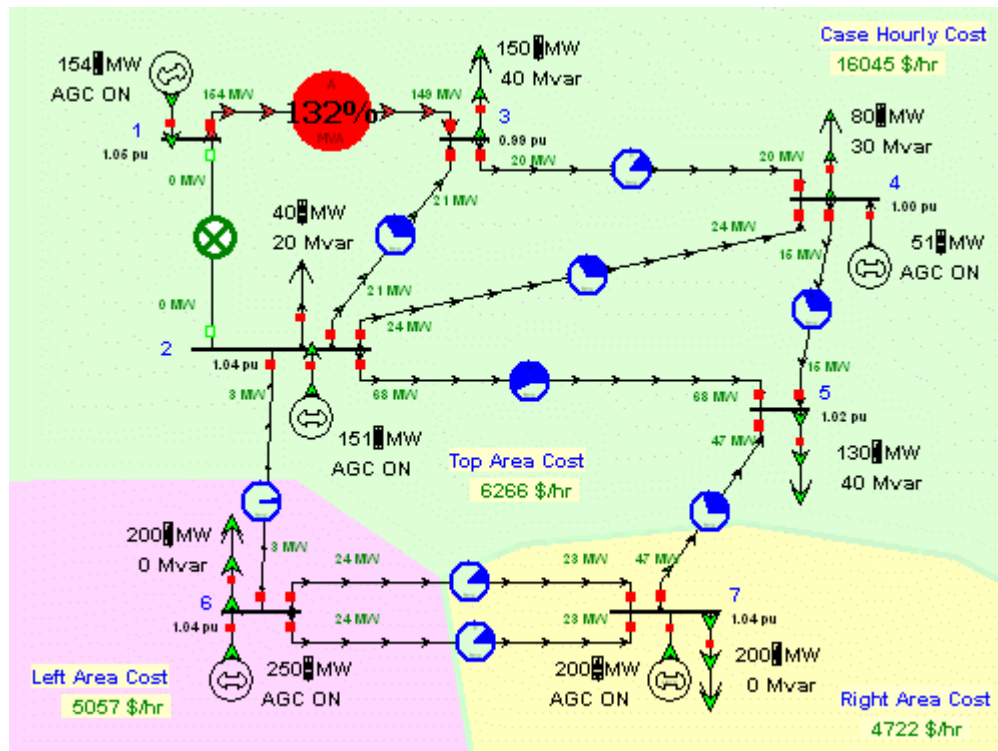
SCOPF Example: Introduction

In this section we introduce an example of using the SCOPF. Consider the seven bus, three area system contained in the file B7SCOPF (included with the PowerWorld Simulator). This case is the same case used in the OPF example, except that the line from bus 2 to 3 has now a 100MVA rating, lines 1 to 2, 1 to 3 and 2 to 5 have a 120MVA rating, and all three areas are initially on OPF control. To initially solve the case using the optimal power flow, select **Primal LP** in the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The solution obtained is shown below. The total case hourly cost is \$16,011 / hr.



B7SCOPF Case Solved using OPF

We are interested in determining an optimal solution that meets security constraints under contingency conditions. In order to show that the current OPF solution does not enforce contingency violations, take line 1 to 2 out of service by clicking in a circuit breaker. Then solve the power flow by pressing the **Single Solution** button. The result indicates that line 1 to 2 is overloaded 32%. The SCOPF algorithm will attempt to move the operating solution such that no contingency violation occurs in the system. Close line 1 to 2 back in service.

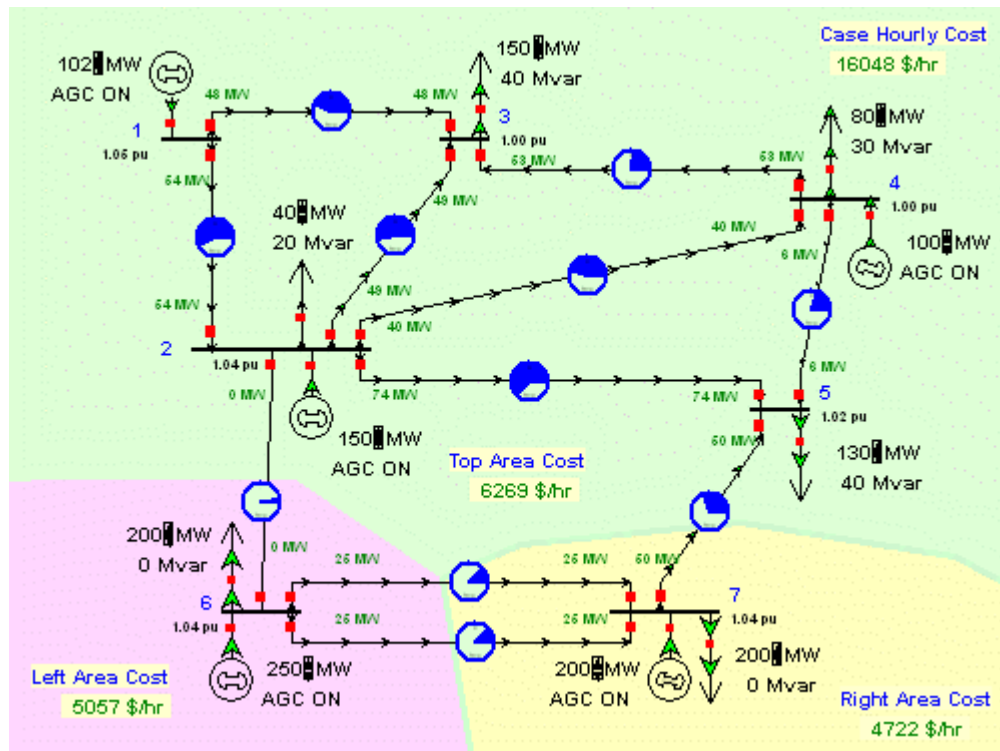


B7SCOPF Power Flow Solution with line 1-2 open

The next step is to specify the contingency conditions that the system should withstand. In order to do that, we access the [Contingency Analysis dialog](#) under **Tools**. Note that we can also access this dialog from the [SCOPF control dialog](#) by selecting **Add Ons > Security Constrained OPF** and pressing the **View Contingency Analysis Form** button on the **Options** page. The B7SCOPF case does not have contingencies associated with it. Insert single line contingencies in the contingency list by pressing the **Auto Insert** button located at the bottom left of the Contingency Analysis dialog. In the [Auto Insert Dialog](#) select the option for **Single Transmission Line or Transformer** and select **Numbers** under the *Identify buses by* field. Select **Do Insert Contingencies**. This will prompt to insert 11 single line contingencies corresponding to all the lines in the system. Select **Yes**. You can now close the Contingency Analysis dialog.

Return to the **Security Constrained Optimal Power Flow Form**. You can set the SCOPF to use the OPF solution as the base by selecting **Solve base case using optimal power flow**.

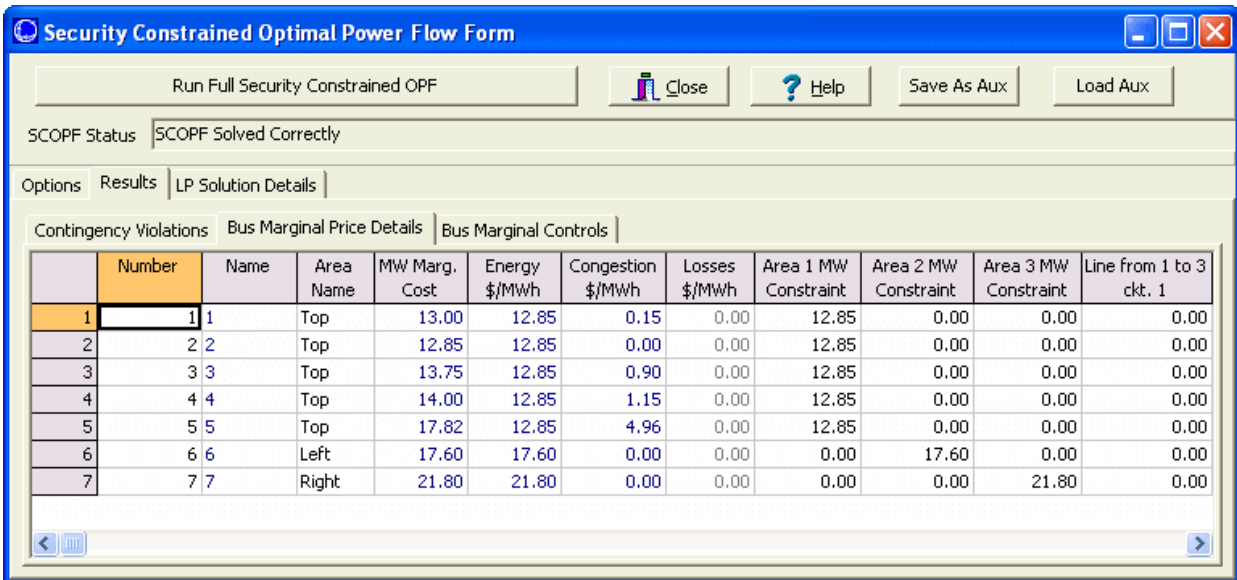
You can now solve the SCOPF by pressing the **Run Full Security Constrained OPF**. This will process the contingency violations and iteratively solve the LP program and the power flow equations to minimize the objective function and enforce equality and inequality constraints. The solution is shown below. The total operating cost is now \$16,048 /hr. The increase in operating cost is due to enforcing security constraints. If a new contingency analysis is performed using the optimal solution as the reference, it will be found that no contingency violations occur for the contingencies in the list, i.e., branch flows are less than (or equal to) 100% in the post contingency condition. Thus, the system meets all the specified constraints. You can analyze the SCOPF results by browsing the information in the SCOPF tabs. Note that the **CTG Constraints** dialog does not show unenforceable constraints, but the branch violation of line 2 to 5 due to the contingency 5 to 7 is now binding.



B7SCOPF Case Solved using SCOPF: The system now meets the contingency constraints.

SCOPF Example: Marginal Prices

Using the SCOPF solution from the previous page, select the **Bus Marginal Price Details** from the **Results** page of the **SCOPF Dialog** to view the detail of the marginal price components. Note in the following Figure that each area constraint contributes equally to the marginal cost of the buses in that area. The binding inequality constraint from bus 2 to 5 makes further contribution to the bus marginal price of buses in area Top.



The screenshot shows the 'Security Constrained Optimal Power Flow Form' window. The 'SCOPF Status' is 'SCOPF Solved Correctly'. The 'Results' tab is selected, and 'LP Solution Details' is the active sub-tab. The 'Bus Marginal Price Details' sub-tab is also visible. The table below displays the marginal price components for 7 buses.

	Number	Name	Area Name	MW Marg. Cost	Energy \$/MWh	Congestion \$/MWh	Losses \$/MWh	Area 1 MW Constraint	Area 2 MW Constraint	Area 3 MW Constraint	Line from 1 to 3 ckt. 1
1	1	1	Top	13.00	12.85	0.15	0.00	12.85	0.00	0.00	0.00
2	2	2	Top	12.85	12.85	0.00	0.00	12.85	0.00	0.00	0.00
3	3	3	Top	13.75	12.85	0.90	0.00	12.85	0.00	0.00	0.00
4	4	4	Top	14.00	12.85	1.15	0.00	12.85	0.00	0.00	0.00
5	5	5	Top	17.82	12.85	4.96	0.00	12.85	0.00	0.00	0.00
6	6	6	Left	17.60	17.60	0.00	0.00	0.00	17.60	0.00	0.00
7	7	7	Right	21.80	21.80	0.00	0.00	0.00	0.00	21.80	0.00

Seven Bus Case SCOPF Bus Marginal Price Details

SCOPF Example: Unenforceable Constraints

Using the same example as above, reduce the MVA rating of line 2 to 5 to 90 MVA. The initial base case LP OPF solution is the same as in the previous example. Consider the same contingency list. When the contingencies are solved using the initial OPF solution as the reference, seven contingency violations need to be removed. The CTG dialog after simulation is presented in the following figure.

	Contingency Name ▲	Element	Value	Scaled Limit	New Value	Error	Included	Marginal Cost	Unenforceable
1	L_00001-00002C1	Br 1 (1) -> 3 (3) CKT 1 at 1	157.31	120.00	-2.76	0.00	NO	0.00	NO
2	L_00001-00003C1	Br 1 (1) -> 2 (2) CKT 1 at 2	154.45	120.00	10.45	0.00	NO	0.00	NO
3	L_00002-00003C1	Br 2 (2) -> 5 (5) CKT 1 at 2	90.73	90.00	68.68	0.00	NO	0.00	NO
4	L_00002-00004C1	Br 2 (2) -> 5 (5) CKT 1 at 2	92.52	90.00	66.18	0.00	NO	0.00	NO
5	L_00006-00007C1	Br 2 (2) -> 5 (5) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
6	L_00006-00007C2	Br 2 (2) -> 5 (5) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
7	L_00007-00005C1	Br 2 (2) -> 5 (5) CKT 1 at 2	127.56	90.00	104.48	14.48	YES	1000.00	YES

CTG Dialog after SCOPF Solution

We note that the constraint from bus 2 to 5 under a contingency from 5 to 7 is unenforceable. There are not enough system controls to enforce the contingency constraint. A \$ 1,000 / hr cost is assigned to unenforceable constraints in this case. The cost of not enforcing constraints can be specified in the [OPF Constraint Options](#) Dialog.

Optimal Power Flow Reserves Overview

Note: The OPF Reserves tool in PowerWorld Simulator is only available if you have purchased the OPF Reserves and OPF add-ons to the base package. To learn more about the OPF Reserves, please read through the information contained in these help files. [Contact PowerWorld Corporation](#) for details about ordering the OPF and OPF Reserves version of Simulator.

In order to prevent load disconnections and loss of stability during normal operation or in the case of unexpected events, power systems should operate with an adequate level of Reserve. Reserve is an ancillary service needed for the successful operation of the system and the electricity market. This service can be obtained in a regulated, mandatory manner, or it can be provided by an Ancillary Services Reserves Market. Simulator OPF Reserves is the tool used to simulate Ancillary Services Reserve Markets.

In the Reserves Market, the generators (and sometimes loads) supply bids to sell the ability to take demand (increase their output) in a fast manner if called to do so. While an energy-only market has only one product, active energy, the Reserves Market includes the energy product and several reserve services: regulating, spinning and supplemental reserve. Similar to electricity markets for energy, which deal with active power, the reserve market focuses only on active power reserves.

The optimal power flow (OPF) algorithm by itself is able to simulate energy-only electricity markets by determining the minimum cost or minimum control change dispatch subject to normal operation constraints. The OPF Reserves considers special OPF Reserve Constraints at the area and zone level, and OPF Reserve Controls provided by generators or loads. OPF Reserves will thus simultaneously co-optimize energy and reserve and maximize total social surplus producing both energy (LMP) and Reserve Marginal Clearing Prices (RMCP).

The commands and options for the [OPF Reserves](#) are integrated with the [OPF tool](#).

OPF Reserves Topics

The commands and options for the OPF Reserves are integrated with the [OPF tool](#). The OPF Reserves option is activated from the [Common Options tab](#) of the OPF Options and Results dialog by checking the **Include OPF Reserve Requirements** box.

The following topics provide details about how the OPF Reserves are integrated into the OPF tool and options and input data can be set:

- [OPF Reserves Controls](#)
- [OPF Reserves Constraints](#)
- [OPF Reserves Objective Function](#)
- [OPF Reserves Results and Pricing](#)
- [OPF Reserves Case Information Display](#)
- [Area and Zone Reserve Requirements Curves](#)
- [Generator and Load OPF Reserve Bids](#)
- [OPF Reserves Example](#)

OPF Reserves Controls

Generators can provide the following types of reserve:

- Regulation Reserve (RR) is provided by online fast units, usually tied to AGC primary control, and capable to regulate the small positive or negative imbalances caused by the random nature of loads.
- Spinning Reserve (SR) is provided by online units and is used to take larger variations of load and losses and unexpected events.
- Supplemental Reserve (XR) is provided by online or offline fast-start units. This type of reserve is used to correct large imbalances caused by contingencies.

Spinning and supplemental reserve are positive quantities. Regulation reserve is a bidirectional control. There are two ways to model regulation controls: As two independent controls: regulation reserve up (RR^+) and regulation reserve down (RR^-). As a single control: in this case the unit provides the same amount of regulating reserve in both directions. In order to tell Simulator whether to use a single bidirectional control for regulating reserve up or down, go to the [OPF Options and Results Dialog](#) and check the **Use a Single Control for Up/Down Regulation** box.

Spinning and supplemental reserves combined together provide Contingency Reserve (CR)

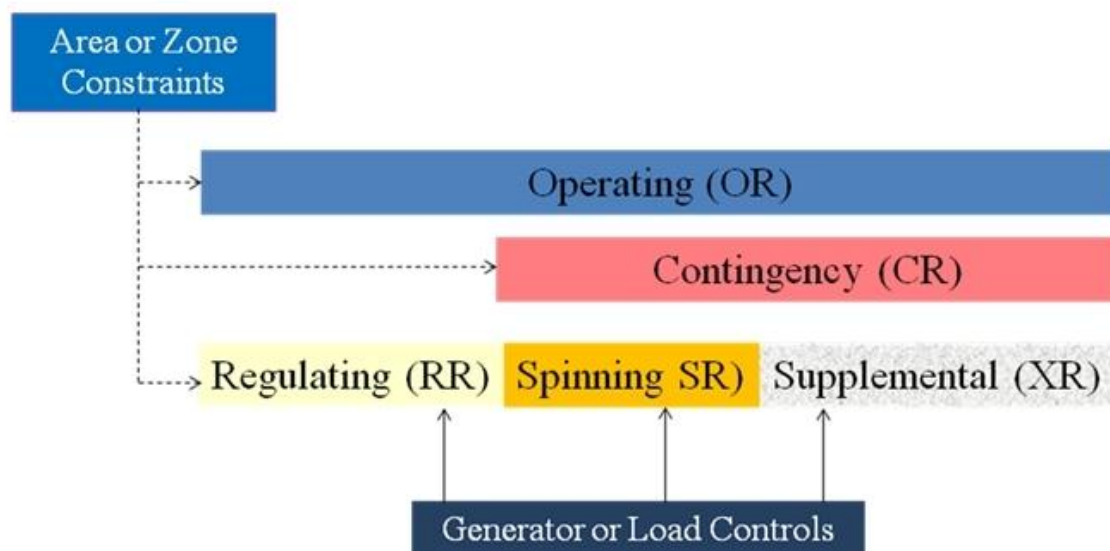
Contingency reserve plus regulating reserve up is called Operating Reserve (OR)

The following relations are then established:

$$RR^+ + SR + XR = OR$$

$$RR^+ + CR = OR$$

Data for the reserve controls can be specified from the [Generator Information dialog](#), the [Load Information dialog](#), or from the corresponding case information displays accessible from the **Model Explorer > Optimal Power Flow > Results > Reserve Results**.



Generator and Load OPF Reserves Bids

Generators and loads can both be providers of reserve services. Similar options exist for both in setting the OPF Reserve Bids parameters. These options can be set for generators on the [Cost tab](#) of the Generator Information dialog and for loads on the [OPF Load Dispatch tab](#) of the Load Information dialog.

OPF Regulating Reserves

These options are only available for generators.

Up Reserve MW - Value in MW of the cleared regulating reserve up. This value is determined by the OPF Reserves solution and is non-enterable by the user.

Down Reserve MW - Value in MW of the cleared regulating reserve down. This value is determined by the OPF Reserves solution and is non-enterable by the user. When a single bidirectional control is used for up and down reserve, the MW value of the cleared Up Reserve and Down Reserve will be the same.

Available for Regulating Reserves - Check this box to declare this unit as a provider of regulating reserves service.

Price [\$/MWh] - This is the bid submitted for the block of regulating up and down reserves.

Maximum MW Increase - This is the size in MW of the block of the regulating reserve up bid submitted to the market. The cleared regulating reserve up cannot be larger than this value.

Maximum MW Decrease - This is the size in MW of the block of regulating reserve down bid submitted to the market. The cleared regulating reserve down cannot be larger than this value.

OPF Contingency Reserves

These options are available for both generators and loads.

Spinning Available - Check this box to declare this unit/load as a provider of spinning reserve service.

Supplemental Available - Check this box to declare this unit/load as a provider of supplemental reserve service.

Reserve MW - Value in MW of the cleared spinning and supplemental reserves. These values are determined by the OPF solution and are non-enterable by the user.

Price [\$/MWh] - This is the bid submitted for the corresponding blocks of spinning and supplemental reserve.

Max MW Increase - For spinning and supplemental reserve this is the size of the reserves block submitted to the market. The cleared reserves cannot be larger than these values.

OPF Reserves Constraints

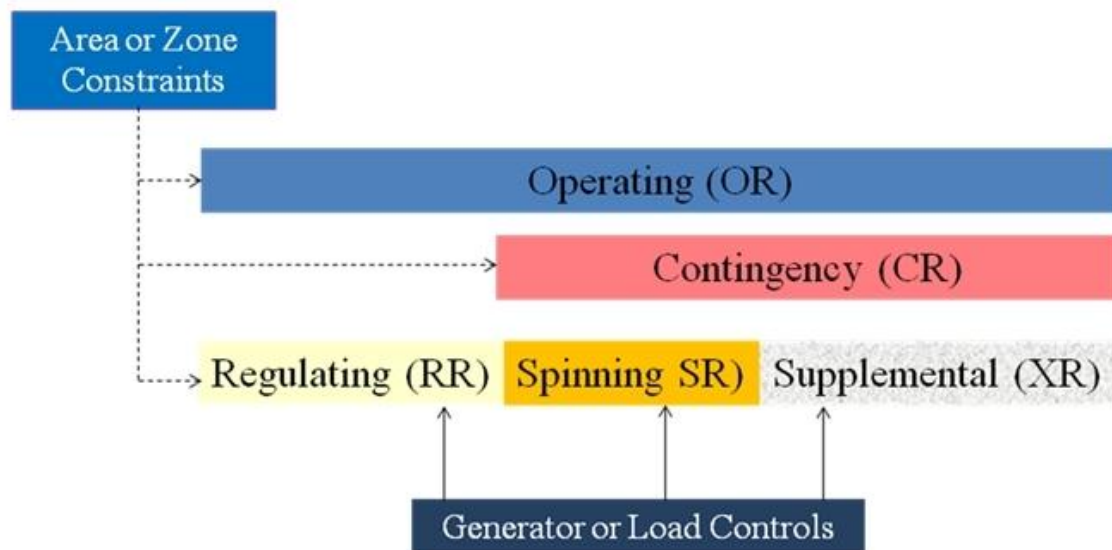
Reserve constraints in the OPF Reserves tool are specified at the area and zone level. Overlapping constraints are handled if an area contains several zones or if a zone spans several areas. For each type of reserve specified, the total reserve of the area or zone must be equal or greater than a certain level of reserve requirement in MW:

- Operating Reserve: $OR \geq OR_{REQ}$
- Regulating Reserve: $RR \geq RR_{REQ}$
- Contingency Reserve: $CR \geq CR_{REQ}$
- Some systems require that a percentage of the contingency reserve be spinning: $SR \geq \%CR$

Reserve requirements are input as incremental demand curves. These can be defined for both areas and zones and are described in more detail in the [Area and Zone Reserve Requirement Curves topic](#).

The reserve requirement demand curves are specified by pairs of (MW,\$/MWh) values that form a stair function. The demand curve should be monotonically decreasing, i.e., while the MW values increase, the \$/MWh values should decrease.

Sometimes, ancillary services markets rules specify a single reserve requirement value and price. For instance, 300MW at \$500/MWh. This requirement can be modeled by a demand curve that contains two points: 0.0 MW at 300 \$/MWh, and 300 MW at 0.0 \$/MWh.



Area and Zone OPF Reserve Requirement Curves

Simulator OPF Reserves requires the user to specify area or zone reserve constraints as reserve demand curves for different types of requirements: operating, regulating and contingency.

Simulator OPF Reserves allows reserve constraints to be specified at the area and zone level or both, including overlapping regions. The data needed to specify reserve constraints at the zone level is identical to that needed for area reserve constraints.

Reserves Requirement Curves are found on the OPF tab of the [Area Information dialog](#) and the OPF tab of either the [edit](#) or [run](#) mode Zone Information dialog.

The reserve requirement incremental demand curves are specified by a set of pairs of (MW,\$/MWh) values that form a stair function. The incremental demand curve should be monotonically decreasing, i.e., while MW values increase, \$/MWh values should decrease.

Sometimes, ancillary services markets rules specify a single reserve requirement and its price. For instance 300MW at \$500/MWh. This requirement would be modeled by a demand curve that contains two points: 0.0 MW at 300 \$/MWh, and 300 MW at 0.0 \$/MWh.

Enforce Operating Reserve Requirement in OPF - Check this box to make the specified operating reserve demand curve be enforced by the OPF Reserves tool.

Enforce Regulating Reserve Requirement in OPF - Check this box to make the specified regulating reserve demand curve be enforced by the OPF Reserves tool.

Enforce Contingency Reserve Requirement in OPF - Check this box to make the specified contingency reserve demand curve be enforced by the OPF Reserves tool.

Results Page - This page shows total results at the area level for different types of reserve quantities: regulating reserve up, regulating reserve down, spinning reserve, supplemental reserve, contingency reserve and operation reserve.

Enforce - Whether a specific type of reserve is enforced for the area. Only regulating, contingency and operation reserve can be enforced at the area or zone level.

Cleared MW - Total reserve cleared in the area, for each reserve type.

Max Reserve MW - Total available reserve in the area for each reserve type.

Hourly Cost \$/hr - Total cost of providing the reserve service in the area for each reserve type.

Price \$/MWh - Reserve Marginal Clearing Price (RMCP) for each reserve type. This price is a result of enforcing the reserve constraints, and thus it is only available for regulation, contingency and operation reserves.

Hourly Benefit \$/Hr - Benefit provided by enforcing reserve constraints at the area level. This benefit is derived from the enforced reserve demand requirements, and it is only available for regulation, contingency and operation reserves.

OPF Reserves Objective Function

The power system should operate exceeding level of the different types of reserves. The benefit associated with a reserve type is equal to the reserve consumer surplus (area below the reserve demand curve and above the equilibrium RMCP). In order to provide reserve, some generators will need to reduce their output to make room for the assigned reserve level. Thus in general, meeting the reserve requirements will result in increased energy prices. In Simulator, the OPF reserves problem is solved using an application of sequential linear programming embedded in the OPF solver. The LP problem consists in maximizing total surplus:

Total Surplus = Benefit - Costs

Where the benefits include:

- Up Regulation Reserve Surplus
- Down Regulation Reserve Surplus
- Contingency Reserve Surplus
- Operating Reserve Surplus

And the costs include:

- Up Regulation Reserve Cost
- Down Regulation Reserve Cost
- Spinning Reserve Cost
- Supplemental Reserve Cost

OPF Reserves Results and Pricing

The results of co-optimization of energy and reserves as implemented in Reserve Markets are:

- Optimal generation dispatch MW set-points and reserve assignments
- Energy locational marginal prices (LMP)
- Reserve market clearing prices (RMCP)

The energy locational marginal prices are different than the prices that would be obtained if reserve requirements are not consider. Usually, LMPs are higher in simultaneous energy and reserve optimization. This is due to some generators having to back-off their output to meet reserve control assignments.

As the bus LMP values correspond to the change in operating cost when an additional MW of active power load is served at that bus, the RMCP correspond to the change in operating surplus when an additional MW of a certain type of reserve is required to be provided at that bus. Each area or zone that specifies reserve requirements will have a RMCP. If particular, the RMCP of buses in overlapping zones and areas will be the result of adding the corresponding RMCP associated with each constraint. The bus components of the RMCP can be accessed from the [Bus MW Marginal Prices Details page](#) in the OPF Options and Results Dialog. RMCP fields will show up automatically for each constraint type that is binding.

OPF Reserves Case Information Display

A special category of [case information display](#) is available from the [Model Explorer](#) to access common OPF reserves data. From the Model Explorer select the **Optimal Power Flow > Results > Reserve Results**. The same set of reserve case information displays can be accessed from the [Add Ons ribbon tab](#) under the **Optimal Power Flow ribbon group > OPF Case Info > OPF Reserves**.

These case information displays include input and output data for the Reserves Markets, such as areas, zones, generator and loads. They have been customized with fields relevant to OPF Reserves. For instance, generator and loads include:

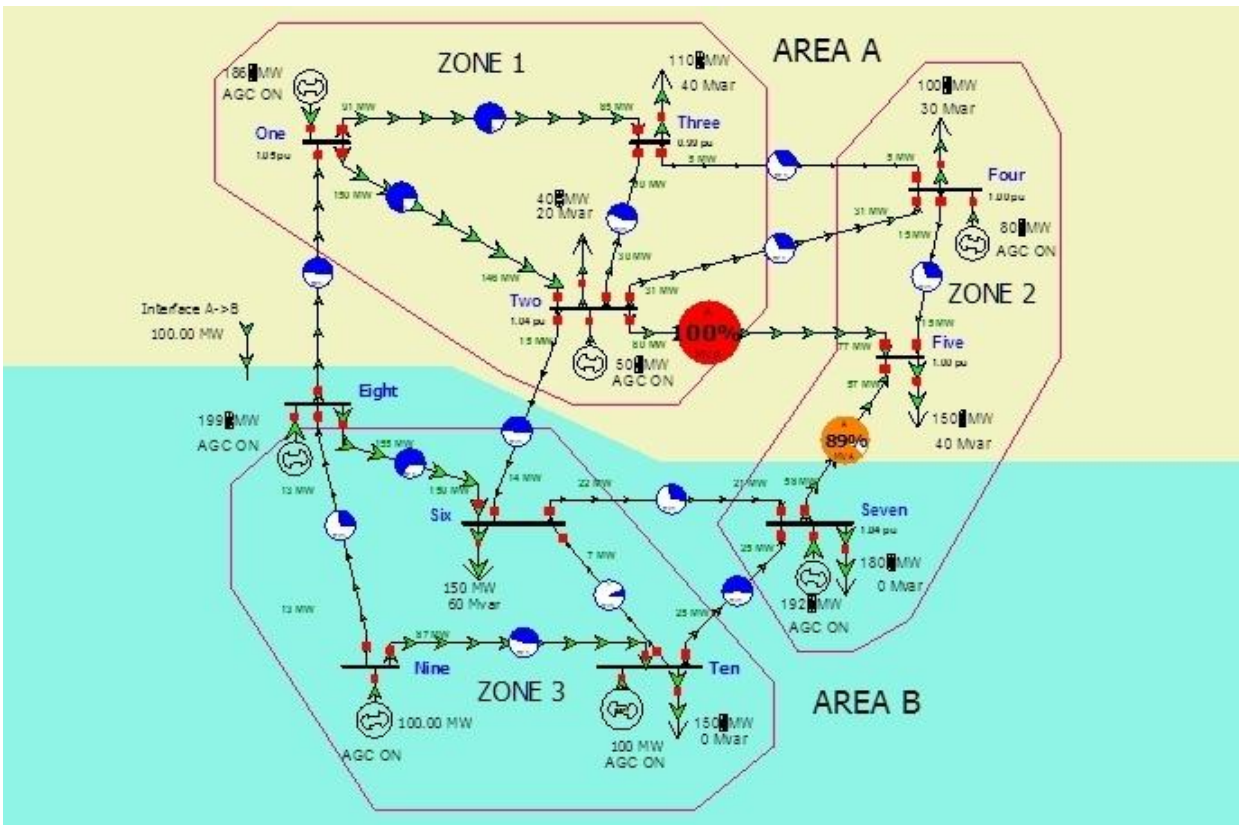
- Availability of each reserve type: regulating, spinning, supplemental
- Maximum reserve available for each reserve type
- Single value bid price for the reserve bid block, for each reserve type
- Resulting RMCP for each reserve type

The area and zone information displays include:

- Enforcement of reserve constraints: regulating, contingency, and operation
- Total cleared reserve for each type
- Total reserve cost and benefit for each type

OPF Reserves Example

In this section, we present an example of the OPF Reserves in action. Consider the 10 bus, 7 generator case contained in the B10Reserve.pwb case (included with PowerWorld Simulator). This case has 2 areas and 3 zones.



First solve the LP OPF without reserves by clicking LP Primal from the [Add Ons ribbon group](#). Check the log to confirm a successful OPF solution. The initial LP OPF solution without reserves presents a binding transmission line as shown in the Figure.

The first step to solving the OPF with reserves is to set up the reserve control availability and reserve bids from the **Add Ons ribbon group > OPF Case Info > OPF Reserves > Generators case information display**. The following parameters, which in this example will be set up in a tabular manner, can also be set in the individual [Generator Information dialog](#). Let us set regulation, spinning and supplemental generator reserve availability as follows:

Gen	Reg. Avail	SPN Avail	SUP Avail
1	YES	NO	NO
2	YES	YES	NO
4	NO	YES	YES
7	YES	NO	YES
8	YES	YES	NO
9	NO	YES	YES
10	NO	NO	YES

Then, let us set up the following reserve blocks and prices (reserve bids) in the same case information diagram.

	REG Max Up	REG Max Down	SPN Max	SUP Max	REG Price	SPN Price	SUP Price
1	50.00	50.00	102.50	29.00	10.00	15.00	10.50
2	60.00	50.00	112.50	34.00	12.00	13.00	9.50
4	70.00	50.00	122.50	39.00	14.00	11.00	8.50
7	80.00	50.00	132.50	44.00	18.00	9.00	7.50
8	90.00	50.00	142.50	49.00	24.00	7.00	6.50
9	100.00	50.00	152.50	54.00	30.00	5.00	5.50
10	110.00	0.00	202.50	59.00	40.00	3.00	4.50

Once the generator and/or load reserve controls have been specified, the second step is to set up the reserve constraints for the control areas and/or zones. Let us set up the following data by accessing each individual [Area Information dialog](#) or [Zone Information dialog](#). Recall, that a single level reserve requirement such as 150 MW @ 400 \$/MWh is set by entering two points in the reserve demand curve: 0.0 MW at 400 \$/MWh and 150 MW at 0.0 \$/MWh.

	Enforce			Requirement		
	OPR	REG	CTG	OPR	REG	CTG
Area A	YES	YES		150 @ 400	80 @ 300	
Area B	YES	YES		90 @ 250	60 @ 150	
Zone 1			YES			65 @ 300
Zone 2			YES			45 @ 140
Zone 3			YES			35 @ 120

Let us now solve the OPF enforcing reserve requirements. Go to the [Add Ons ribbon tab](#) and choose [OPF Options and Results](#) in the Optimal Power Flow ribbon group. Check the box **Include OPF Reserve Requirements**. Click the Solve LP OPF button from the dialog or the Primal LP button from the Add Ons ribbon tab.

The OPF Reserves, will enforce reserve constraints for each reserve type and for each area and zone whose Enforce Field is set to *YES*. In addition, OPF Reserve will observe the following constraints:

- Transmission line limits
- Generator MW Max and Min limits for the total energy plus reserve controls
- Area scheduled interchange. For this case, there is a 100 MW export from Area A to Area B.

To explore the results, go to the Generators page in [OPF Reserve Results](#) in the Model Explorer. The results include generator MW dispatch and assignment of all the reserve services. Note that in this example, some of the generators have reached their reserve bid limits for certain units and types of reserve.

OPF Reserve Requirements Results														
Records Geo Set Columns Options														
Generators Loads Areas Zones Super Areas														
	Number of Bus	Name of Bus	ID	REG Avail.	SPN Avail.	SUP Avail.	Min MW	Min Reg MW	Gen MW	REG MW Up	REG MW Down	SPN MW Up	SUP MW Up	Max Reg MW
1	1	One	1	YES	NO	NO	50.00	0.00	160.00	50.00	50.00	0.00	0.00	250.00
2	2	Two	1	YES	YES	NO	50.00	0.00	80.00	30.00	30.00	65.00	0.00	300.00
3	4	Four	1	NO	YES	YES	50.00	0.00	74.72	0.00	0.00	0.00	5.00	200.00
4	7	Seven	1	YES	NO	YES	0.00	0.00	198.60	43.00	50.00	0.00	40.00	300.00
5	8	Eight	1	YES	YES	NO	150.00	0.00	191.68	0.00	10.00	0.00	0.00	300.00
6	9	Nine	1	NO	YES	YES	0.00	0.00	100.00	0.00	0.00	0.00	0.00	200.00
7	10	Ten	1	NO	NO	YES	0.00	0.00	99.87	0.00	0.00	0.00	7.00	1000.00

Results can be explored at the area or zone level by accessing the individual Area Information dialog or Zone Information dialog, going to the OPF tab, and the Results subtab of the [Reserve Requirement Curves](#). For instance, the Results for Area A are as shown in the next Figure. For each area, the results page shows the enforcement of different types of regulation, the total cleared reserve, the total available reserve, the reserve cost per hour, the reserve price, and the hourly benefit, which corresponds to the area below the demand curve and above the equilibrium RMCP level.

Reserve Requirement Curves						
Operating	Regulating	Contingency	Results			
Enforce	REG +	REG--	SPN	SUP	CTG	OPR
	YES	YES	NO	NO	NO	YES
Cleared MW	80.00	80.00	65.00	5.00	70.00	150.00
Max Reserve MW	110.00	100.00	235.00	39.00	274.00	384.00
Hourly Cost \$/hr	860.00	860.00	845.00	42.50	887.50	1747.50
Price \$/MWh	143.00	12.00			-201.00	-131.00
Hourly Benefit \$/hr	24000.00	24000.00			0.00	60000.00

The Results tab of the [OPF Options and Results dialog](#) shows the details of Bus MW marginal prices, including energy, congestion and losses, and the reserve prices associated with the area or zone reserve constraints. The corresponding linear programming details will list among the LP variables the reserve controls specified by the user. The LP tableau will include specific rows for the enforced reserve constraints at the area and zone level, etc. Reserve results can be visualized as any other generator, load or bus object field in Simulator.

Available Transfer Capability (ATC) Analysis

The ATC Analysis tool is only available if you have purchased the ATC add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the ATC version of Simulator.

Available Transfer Capability (ATC) analysis determines the maximum incremental MW transfer possible between two parts of a power system without violating any specified limits. Most often, this transfer is between two areas in the system, but can be customized to specific groups of system devices.

Simulator's ATC analysis makes use of several tools that are available elsewhere in Simulator. These include:

- [Power Transfer Distribution Factors \(PTDFs\)](#): determine the linear impact of a transfer (or changes in power injection) on the elements of the power system.
- [Line Outage Distribution Factors \(LODFs\)](#): determine the linear impact of a line outage on the elements of the power system.
- [Contingency Analysis](#): studies the impact of a list of contingencies on the power system.
- [Limit Monitoring Settings](#): control which elements of the system are monitored for limit violations.

You do not directly use these other tools when using Simulator's ATC analysis tool, but Simulator uses the settings and algorithms in the background to determine ATC. Thus, it is helpful to be knowledgeable on their use, as it will help you in interpreting the results of an ATC analysis.

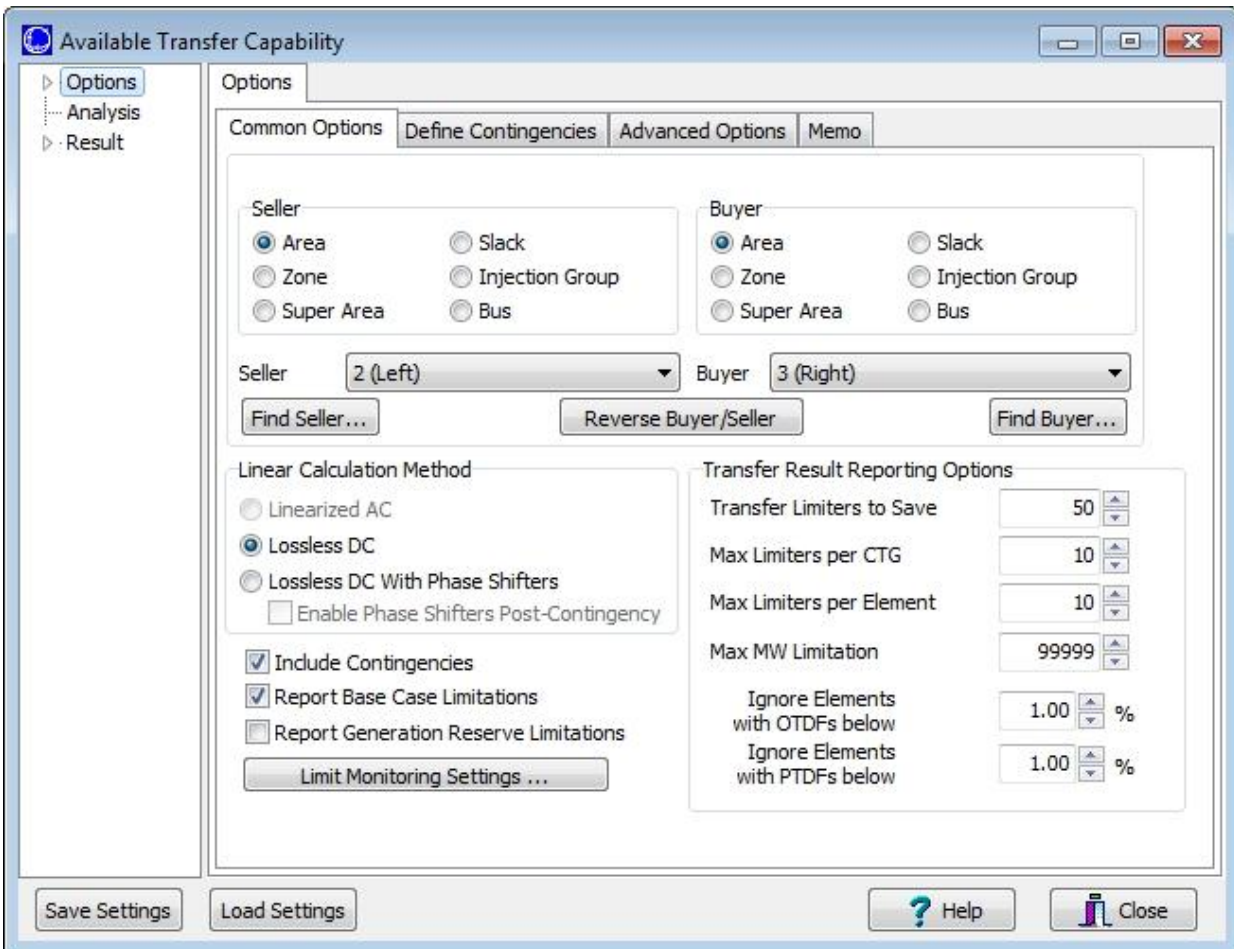
- Simulator provides three methods of determining the ATC for a transfer direction. See : [ATC Solution Methods](#)

For information on how to use the Simulator ATC tool, see [Available Transfer Capability Dialog](#).

Available Transfer Capability Dialog

The Available Transfer Capability dialog provides an interface for performing and viewing the results of [Available Transfer Capability Analysis](#). ATC analysis is typically only done on the present power system state or scenario (called Single Scenario ATC Analysis). Click [here](#) for information on performing [multiple scenarios](#) analysis.

To open the ATC dialog, go to the [Add Ons](#) ribbon tab and select **Available Transfer Capability (ATC)** from the **ATC** ribbon tab. The dialog opens to the Single Scenario ATC Analysis version with the Options page, Common Options sub-tab visible.



The ATC dialog is divided in three pages: [Options](#), [Analysis](#), and [Result](#). If **Analyze Multiple Scenarios** has been checked on the [Advanced Options sub-tab](#) of the [Options](#) page, the layout of the ATC dialog is slightly different—see [Multiple Scenarios ATC Dialog](#).

In addition, the following buttons can be used to save/load ATC analysis settings:

Save/Load Settings

ATC Analysis settings can be saved by selecting **Save Settings** on the ATC dialog. This allows you to repeat the analysis without having to reconfigure the settings. Select **Load Settings** to retrieve previously saved settings.

ATC Dialog: Options

The Options page is available on the [Available Transfer Capability dialog](#). This page is subdivided into four sub-tabs:



[Common Options](#)

Define Contingencies

This sub-tab is similar to the [Contingencies Tab](#) of the [Contingency Analysis Dialog](#). The user can insert, auto-insert, define and/or delete contingency records. See [Contingency Analysis](#) for detailed information on defining contingencies.

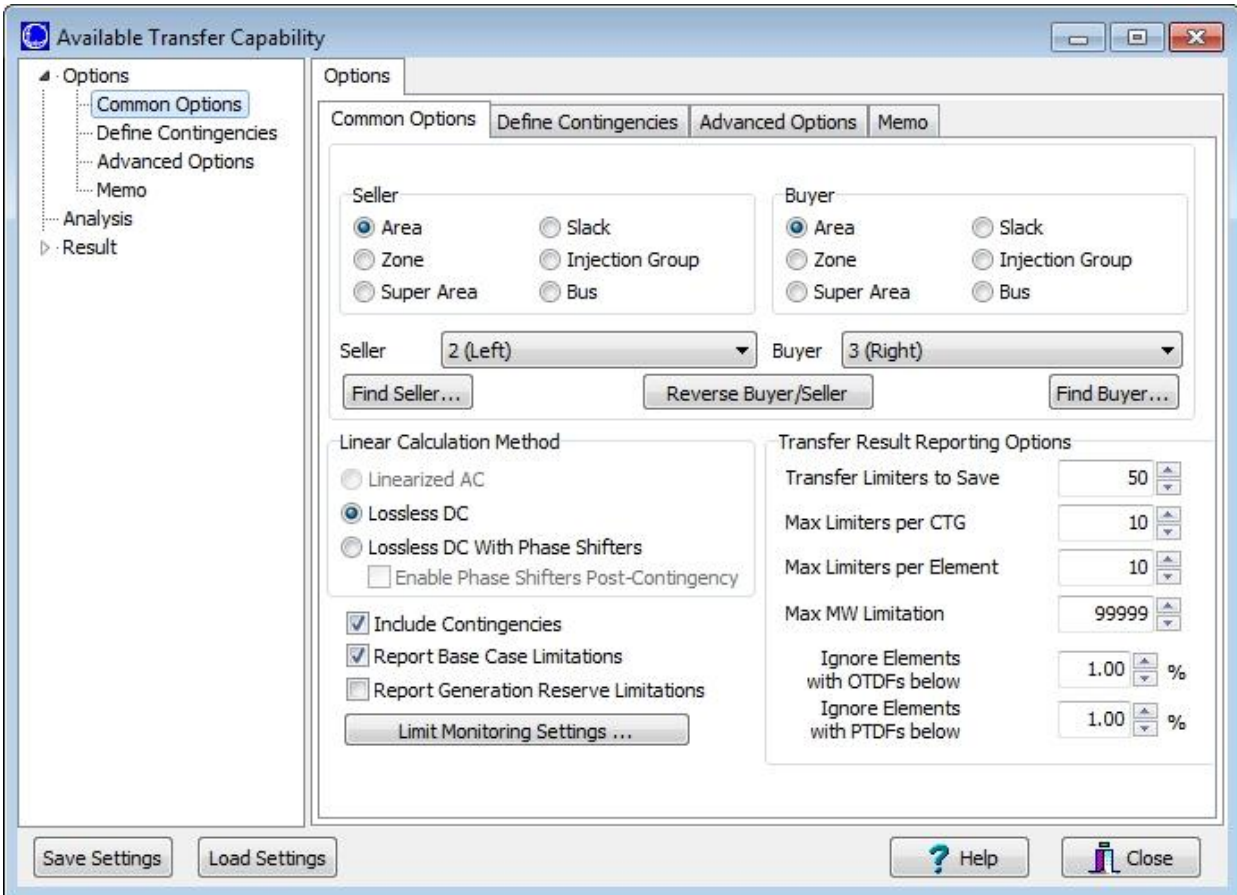
[Advanced Options](#)

Memo

Notes related to the ATC analysis, or anything else for that matter, can be entered on this sub-tab. These notes will be saved to an auxiliary file if choosing to save the ATC settings.

ATC Dialog Options: Common Options

The Common Options sub-tab is found on the [Options](#) page of the [Available Transfer Capability dialog](#).



The following parameters can be set on the Common Options page:

Seller Type, Buyer Type

For the [ATC Solution Method of Single Linear Step \(SL\)](#), transfer limits can be calculated for transfers between combinations of areas, zones, [super areas](#), [injection groups](#), buses, or to the system slack bus. For the iterated linear step ATC Solution Methods ([IL](#) and [IL then full contingency solution](#)), transfer limits can be calculated for transfers between areas, [super areas](#) or [injection groups](#) only. Use the seller type and buyer type options to indicate the type of the selling and purchasing entities.

Seller, Buyer

These dropdown boxes allow you to select the selling and buying entities. The contents of each depend on the Seller/Buyer Types selected. Clicking the **Find Seller** and **Find Buyer** buttons allows you to use Simulator's [Advanced Search Engine](#) to locate the desired entities.

The Seller defines where power is injection into the system and the Buyer defines where power is taken out of the system. The transfer is simulated by increasing generation or decreasing load in the Seller and decreasing generation or increasing load in the Buyer. The [ATC Analysis Methods - Solution Methods](#) go into greater detail about how the impact of the transfer is actually determined. When areas, zones, or super areas are selected to be either the Seller or Buyer, only on-line

generators in the defined group that are on AGC control are allowed to participate in the transfer. Each generator participates in the transfer in proportion to its participation factor. When injection groups are selected to be either the Seller or Buyer, only on-line generators and loads defined with the injection group are allowed to participate in the transfer. Each element participates in the transfer in proportion to the participation factor defined with its [participation point](#) in the injection group. When a single bus is selected as the Seller or Buyer then all of the power injection change needed for the transfer comes from that bus. It does not matter if there is a generator or load at that bus. When the slack is selected for the Seller, then all of the power injection change needed for the seller side of the transfer comes from the island slack bus for the island in which the Buyer belongs. When the slack is selected as the Buyer, then all of the power injection change needed for the buyer side of the transfer comes from the island slack bus for the island in which the Seller belongs.

Generator and load MW limits are not enforced regardless of the type of buyer and seller. Error messages will be displayed if the Seller and Buyer are not both completely contained within the same electrical island.

Reverse Buyer/Seller Button

Click this button to reverse the direction currently shown. The buyer becomes the seller, and the seller the buyer.

Linear Calculation Method

The ATC analysis tool can use either a **Lossless DC** or **Lossless DC with Phase Shifters** calculation method for obtaining the ATC results.

If you select the Lossless DC option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals.

The Lossless DC with Phase Shifters method, a modification to the lossless dc approximation, takes into account phase shifter operation. It is especially useful when the ATC tool continually reports overloads on branches that obviously will not overload because of the operation of a phase shifting transformer. This method assumes that the phase shifter angles may change to any value to ensure that the line flow on those lines does not change.

The Linearized AC method is not yet available.

Enable Phase Shifters Post-Contingency

This option becomes enabled when you choose the **Lossless DC with Phase Shifters** linear calculation method. It allows you to choose whether or not phase shifter control should be enforced during post-contingency ATC calculations. When this option is checked, it is assumed that phase shifter angles may change to keep the line flow from changing after the contingency has been applied. Otherwise, it is assumed that phase shifters only maintain the flow in the pre-contingency state and the angles do not change in the post-contingency state. Base case (pre-contingency) transfer limiters will still be determined with phase shifters enforced, regardless of the setting of this option when the Lossless DC with Phase Shifters option is checked.

Include Contingencies Check Box

Check this box to include contingencies (inserted or loaded using the [Contingency Analysis Tool](#)) in the ATC analysis. Note that minimizing the number of contingencies considered greatly improves solution speed as well as computer memory requirements for doing ATC Analysis. Therefore, be careful in choosing which contingencies to use with the ATC tool. Setting the **Skip** field to **YES** for a particular contingency will exclude it from the ATC analysis.

The ATC tool uses the same contingency records as the contingency analysis tool. Contingency results may be invalid following ATC analysis. If necessary, save contingency results to an auxiliary file prior to running the ATC analysis.

Contingency elements have an associated [Status](#) that determines under what conditions a particular action will be implemented. When ATC values are calculated using linear methods, contingency actions are not actually implemented. Because contingency actions are not actually implemented, *POSTCHECK* actions cannot be applied after other actions have been performed and the power flow has been solved. Instead, *POSTCHECK* actions act as *CHECK* actions during [linear ATC analysis \(Single Linear Step Method\)](#). During linear ATC analysis whether or not model criteria are met for *CHECK* and *POSTCHECK* contingency actions will be determined at the base transfer level rather than at the transfer level that determines the limiting transfer. The base transfer level is the transfer level at which the linear ATC values are actually determined. In most cases this will be the zero transfer level, but this can be different if using one of the iterated ATC methods.

Report Base Case Limitations Check Box

When checked, the ATC tool will report transfer limitations from the base case (pre-contingency).

Report Generation Reserve Limitations Check Box

When checked, the ATC tool will report transfer limitations from generation reserve.

Generator and load MW limits are not enforced when calculating linear sensitivities used to determine the transfer MW limits. However, checking this option provides an indication of when power transfers are limited based on generator maximum or minimum MW limits or the amount of load that is on-line. When this option is checked and there are limitations to how much power can be transferred because of max/min generator limits or the on-line load, additional Transfer Limiters will be present in the [results](#). The Limiting Element will indicate either the Buyer or Seller and the Transfer Limit in MW will be the amount of transfer that can be achieved before the Buyer or Seller exceeds some limitation to generation or load. When considering reserve limits for the Seller, generators will be included in the reserve limit by taking the difference between the generator maximum MW limit and the present output and loads will be included in the reserve limit by the present amount of on-line load. When considering reserve limits for the Buyer, generators will be included in the reserve limit by taking the difference between the present output of the generators and the minimum MW limit and loads will be included by an arbitrary value of 9999.9 MW. Loads are included in this manner because maximum and minimum limits on loads are not currently used in the ATC tool.

For example, assume that an area that currently has 200 MW on-line and a total maximum MW output of 250 MW for all on-line generators that are on AGC control is defined as the Seller. A 50 MW generation reserve limitation will be reported for this area. As another example, assume that an injection group currently has 50 MW of on-line generation with a total maximum MW output of 150 MW and 50 MW of on-line load is defined as the Seller. A 150 MW reserve limitation will be reported for this injection group.

No generation reserve limitations are reported when the seller or buyer type is a bus or the slack.

If using one of the iterated methods, this option will be ignored except during the first single linear step. Reserve limitations may be reported from this step, but limitations of this type will not be iterated on during the individual iteration process. If choosing to enforce generator MW limits appropriately with the type of transactors that are defined, reserve limits will still be reported, regardless of this option setting, if they are hit during any step of the iterative process when ramping is occurring. See the detailed descriptions of either the [Iterated Linear Step \(IL\)](#) or [Iterated Linear Step \(IL\) then Full CTG Solution](#) method for more information about how reserve limits are checked during the iterated methods.

Limit Monitoring Settings Button

Click this button to open the [Limit Monitoring Settings Dialog](#). Limiting Elements will only be reported for those elements that are selected to be monitored. Note that minimizing the number of monitored power system elements greatly improves solution speed as well as computer memory requirements for doing ATC Analysis.

[Monitoring Exceptions](#) can be used to include or exclude specific monitored elements on a contingency-by-contingency basis.

Transfer Result Reporting Options

The options in this section of the ATC dialog are discussed in detail in the topic on [Transfer Result Reporting Options](#).

ATC Dialog Common Options: Transfer Result Reporting Options

These options are found in the Transfer Result Report Options section on the [Common Options](#) sub-tab of the [Options](#) page found on the [Available Transfer Capability dialog](#).

Transfer Limiters To Save

This value tells Simulator how many total Transfer Limiters to save. Simulator will save those Transfer Limiters with the lowest Transfer Limitation. An explanation of a Transfer Limiter follows:

During Linear ATC Analysis, Simulator determines the Transfer Limitation (See [Available Transfer Capability Analysis](#)) for each transmission line and interface during each contingency and the base case. From this Simulator develops a list of Transfer Limiters. A Transfer Limit contains three key pieces of information:

- Transfer Limit in MW
- Limiting Element: Transmission branch (or interface) that causes the limit
- Limiting Contingency: Contingency that is applied to cause the Limiting Element to overload (if it is a limit without any contingency applied, then the contingency will say *Base Case*)

Thus if we are monitoring 1000 transmission lines during 99 contingencies plus 1 base case, there would be 100,000 Transfer Limiters calculated. We are not concerned with all 100,000 limitations, therefore, only the limitations with the smallest Transfer Limit in MW are reported.

Max Limiters per CTG

When analyzing a long list of contingencies, the worst transfer limitations may all occur during the same contingency. Set this value to limit the number of Transfer Limiters saved that are associated with a given contingency.

Max Limiters per Element

When analyzing a long list of contingencies, the worst transfer limitations may all be overloads of the same limiting element. Set this value to limit the number of Transfer Limiters saved that are associated with a given Limiting Element.

Max MW Limitation

This value defines the maximum transfer to report between the buyer and seller. Simulator will compute the ATC analysis results until the transfer amount reaches the value in this field and only report the results meeting the other reporting criteria up to this MW limitation.

When using one of the iterated methods, this option is only applied during the initial Single Linear Step portion of the method. Once individual limiters are being iterated on, this option will be ignored.

Ignore Elements with OTDFs below

Simulator will not report Transfer Limitations for elements with [OTDF](#) values less than this user-specified value. The default value is 0.5%, meaning that for a 100 MW transfer, there would be only a 0.5 MW increase in flow on the Limiting Element. The OTDF cutoff is only used with transfer limitations that include contingencies.

Ignore Elements with PTDFs below

Simulator will not report Transfer Limitations for elements with [PTDF](#) values less than this user-specified value. The default value is 0.5%. The PTDF cutoff is only used with transfer limitations that do not include contingencies, i.e. base case limitations.

For interfaces that include contingent elements, no additional contingencies will be studied on top of the contingencies that are part of the interface. The cutoff value to use for the OTDF is the minimum of the OTDF and PTDF cutoff values.

The transfer limitation functions involve dividing by the [PTDF](#) or [OTDF](#) values for each branch or interface to calculate the Transfer Limit. This leads to two facts:

- The accuracy of the transfer limitation is less for lines that have very small [PTDF](#) or [OTDF](#) values.
- A very small [PTDF](#) or [OTDF](#) value means that the transfer has very little impact on the line.

These two facts often result in Linear ATC analysis reporting inaccurate transfer limitations for lines that are largely unaffected by the transfer. It is not uncommon to have a transfer limitation report an extremely negative transfer limit (e.g. -1.9E28 MW). A branch which is overloaded by a very small percentage, but which has a very small OTDF value often causes this. If the OTDF value is 0.001%, then a branch overloaded by 1 MW will result in a transfer limitation of -100,000 MW.

This motivates the usefulness of ignoring elements with small [PTDFs](#) and [OTDFs](#).

All of these options work together to determine the list of limiters that shows up in the results. The [Limit Monitoring Settings](#) determine elements that will be considered for monitoring. The [list of defined contingencies](#) determines the contingencies. Every combination of monitored elements and contingencies will be tested. [Monitoring exceptions](#) can also be used to include or exclude monitored elements on a contingency-by-contingency basis. If the PTDF and/or OTDF cutoffs, as appropriate, are met for the monitored element/contingency pair, the monitored element and contingency are then checked against the **Max Limiters per Element** and **Max Limiters per CTG** conditions.

For each monitored element, a maximum, X, of **Max Limiters per Element** is kept during the initial processing. Only the most limiting X limiters for that element are retained. The transfer limiter with the highest transfer limit value will get pushed out of the list for an element if a new combination of monitored element/contingency is more limiting than what is currently in the list. For each contingency, a maximum, Y, of **Max Limiters per CTG** is kept during the initial processing. Only the most limiting Y limiters that for contingency are retained. The transfer limiter with the highest transfer limit value will get pushed out of the list for a contingency if a new combination of monitored element/contingency is more limiting than what is currently in the list. Once all monitored element/contingency pairs have been processed, all of the individual lists for limiting elements and contingencies are merged into one big list that is sorted by transfer limit, limiting element, and contingency. This sorted list is then traversed in the order of increasing transfer limit to create the final list that enforces the **Max Limiter per Element**, **Max Limiters per CTG**, **Transfer Limiters to Save**, and **Max MW Limitation** conditions. If there are no monitored element/contingency pairs that are below the Max MW Limitation, the monitored element/contingency pair with the lowest transfer limit value will be the only record reported in the results.

The following is an example of how this process works:

Max Limiters per CTG = 2, Max Elements per Limiter = 1

Limiting Element Lists

Line A contains transfer limiter for CTG B with ATC = 500

Line B contains transfer limiter for CTG B with ATC = 50

Line C contains transfer limiter for CTG X with ATC = 125

Line D contains transfer limiter for CTG A with ATC = 750
Line H contains transfer limiter for CTG B with ATC = 10
Line Z contains transfer limiter for CTG D with ATC = 10

Contingency Lists

CTG A - Line B with ATC = 1500, Line C with ATC = 1750
CTG B - Line H with ATC = 10, Line Z with ATC = 25
CTG C - Line D with ATC = 1000
CTG D - Line Z with ATC = 10
CTG X - Line C with ATC = 125

Now combine the lists and sort based on ATC, limiting element, and contingency. Duplicate entries can exist in the list because the same limiting element/contingency pair can be stored in both the limiting element list and the contingency list.

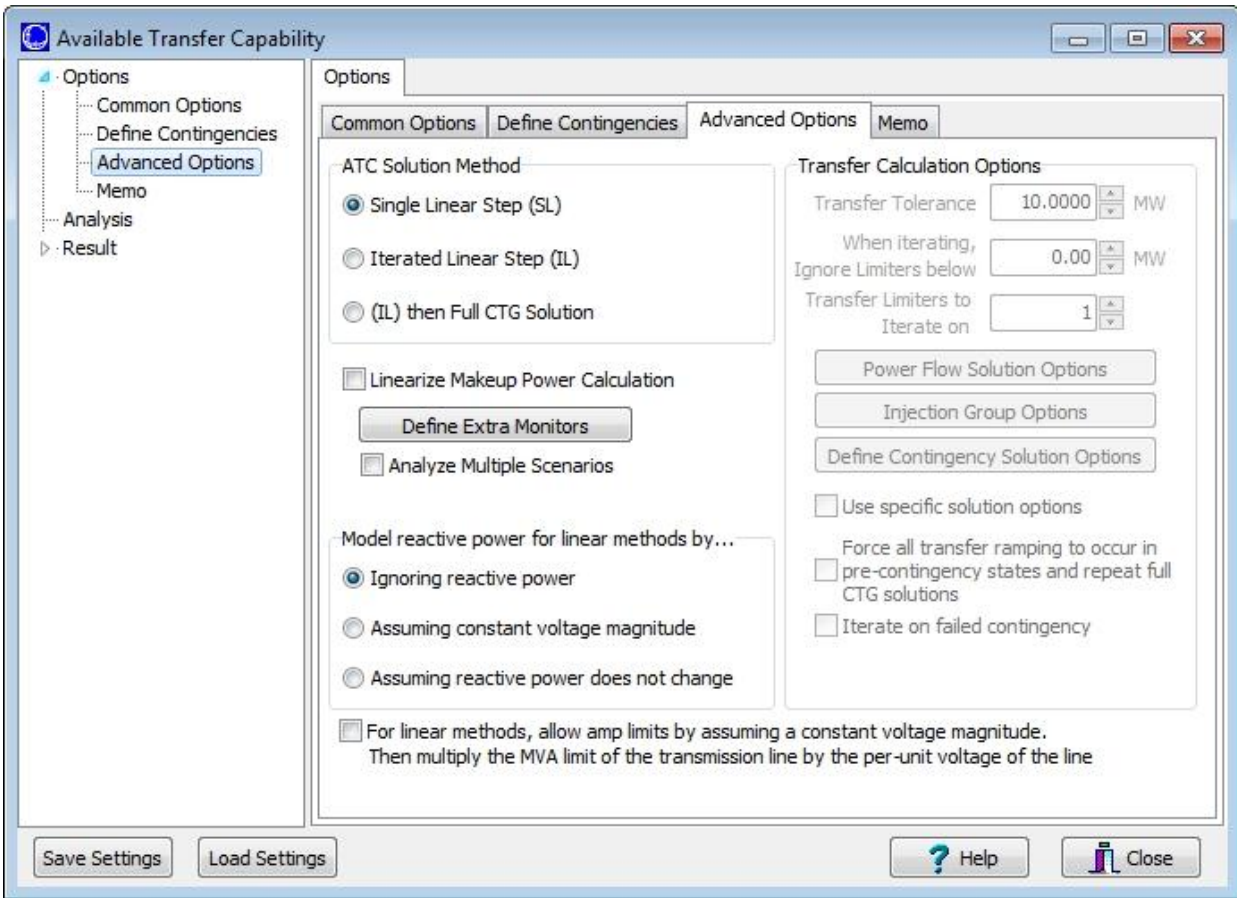
10 Line H/CTG B
10 Line H/CTG B
10 Line Z/CTG D
10 Line Z/CTG D
25 Line Z/CTG B
50 Line B/CTG B
125 Line C/CTG X
125 Line C/CTG X
500 Line A/CTG B
750 Line C/CTG A
1000 Line D/CTG A
1500 Line B/CTG A
1750 Line C/CTG A

Now go through the sorted list above to produce the final results list that abides by all conditions.

10 Line H/CTG B
10 Line Z/CTG D
50 Line B/CTG B
125 Line C/CTG X
1000 Line D/CTG A

ATC Dialog Options: Advanced Options

The Advanced Options sub-tab is found on the [Options](#) page of the [Available Transfer Capability dialog](#).



The following parameters can be set on the Advanced Options sub-tab:

ATC Solution Method

One of the [solution methods](#) for determining the ATC of a specified transfer direction.

If using either the Iterated Linear Step (IL) or (IL) then Full CTG Solution method, and the seller and buyer are either areas or super areas, and the automatic generation control method for the case is not set to area/super area control, an error message will be produced and the analysis will not continue. In the ATC analysis, transfers between areas are implemented by adding a transaction between areas which requires them to be on area interchange control in order to meet the ACE requirement. If the case was just switched to this control method without the areas already being on area control, the dispatch for the case could drastically change. The burden is left on the user to ensure that control methods are set up correctly. If the case is not already on area interchange control, but one of the island-based methods, and putting it on area control would drastically change the case, the ATC analysis can still be done between areas if injection groups are created to represent the areas and the analysis is then done using these injection groups as the seller and buyer.

Linearize Makeup Power Calculation

When doing the linear ATC calculations, any contingencies that cause a MW change (e.g. dropping generation or changing load) require makeup power to compensate for the change in MW. When this option is checked, a precalculation is done at the beginning of each linear step calculation that determines the impact of makeup power on line flows. When a particular contingency is processed, the effect of makeup power on all monitored lines will be determined by multiplying the precalculated effect of makeup power on a line by the total amount of makeup power needed for the contingency. This calculation is slightly faster than calculating the impact of the makeup power separately for each contingency, but it will not take into account the fact that larger amounts of required makeup power may cause generators to hit their limits.

Define Extra Monitors

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize Extra Monitors. Click the **Define Extra Monitors** button to open the [ATC Extra Monitors dialog](#). The flow on the extra monitors will be reported for each transfer limiter determined.

Analyze Multiple Scenarios

Check Analyze Multiple Scenarios to perform ATC Analysis on several scenarios. See [ATC Dialog for Multiple Scenarios](#) for more information on Multiple Scenario Analysis.

Model Reactive Power for Linear Methods By...

The linearized methods used in the ATC tool are based only on the changes in real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations. The chosen assumption allows a MW limit to be calculated from the defined MVA limit for use in determining the ATC values. Before applying the option for handling reactive power, the specified MVA limit for a branch is used to determine the starting MVA limit. The [Limit Monitoring Settings](#) can cause an adjustment the defined MVA limit based on the **Branch Percentage** specified. The modified limit that is used in the calculations is then:

$$MVALimit = DefinedMVALimit * BranchPercentage$$

The choices for adjusting the MVA Limit for reactive power are then:

Ignoring reactive power – reactive power is completely ignored. The MW limit of a line is assumed to be the modified MVA limit of the line:

$$MWLimit = MVALimit$$

Assuming constant voltage magnitude – reactive power is adjusted to hold voltage magnitudes constant. This option uses the intersection of a branch's operating circle and limiting circle to adjust the limit. The operating circle defines a circle of valid MW and Mvar values for a branch as a transfer takes place across the system. The limiting circle has a radius equal to the modified MVA limit of the branch.

Assuming reactive power does not change – reactive power is held at pre-ATC solution levels. The MW limit for each line is calculated from the MVA limit and the reactive power at the pre-ATC solution level:

$$MWLimit = \sqrt{(MVALimit)^2 - (Mvar)^2}$$

For Linear Methods, Allow Amp Limits by Assuming a Constant Voltage Magnitude

If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the base case full AC load flow operating point just prior to the ATC linear calculations. This option will only be used if the [Limit Monitoring Settings Option of Treat Line Limits as Equivalent Amps](#) is checked.

This option will cause the ATC tool to internally use a modified MVA limit which is equal to the user-specified MVA rating multiplied by the actual terminal voltages of the line. This results in a modified MVA rating which is higher than the user-specified MVA rating when the voltage on the line is higher than nominal and lower when the voltage is lower than nominal. For purposes of calculating the ATC values, a MW limit is determined from the MVA rating based on how this option is set and how the reactive power is modeled.

Ignoring reactive power

$$MWLimit = MVALimit * V$$

Assuming reactive power does not change

$$MWLimit = \sqrt{(MVALimit * V)^2 - (Mvar)^2}$$

where V is the voltage at the end of the branch from which the limit is taken. The Transfer Limit is calculated at each end of a branch and the most limiting value is reported as the Transfer Limit for a given branch.

Transfer Calculation Options

Note: The Transfer Calculation Options section is disabled if the [Single Linear Step](#) Solution Method is selected.

Transfer Tolerance

This is the tolerance used during the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The iterated methods will finish if the transfer step amount for the next iteration is less than this tolerance. The default is 10 MW.

When Iterating, Ignore Limiters below

The iterated techniques perform additional analysis on individual transfer limitations in the order of increasing Transfer Limit in MW. When using one of the iterated techniques, transfer limitations with Transfer Limit values below a certain threshold can be ignored for the iterated portion of the process. Transfer limitations with Transfer Limit values equal to or below the value set with this option may still appear in the results, but they will not be iterated on individually.

Once an individual limiter is being iterated on, this value will no longer be used. This means that during the individual iteration process, the transfer level for a limiter can fall below this threshold.

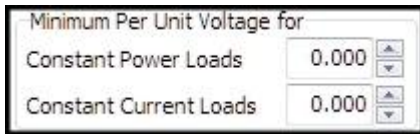
Transfer Limiters to Iterate on

This is the number of transfer limiters to iterate on in the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The default is 1. The transfer limiters actually iterated on are determined by selecting the number of limiters set in this option in ascending order of Transfer Limit with the Transfer Limit at or above the value set in the **When Iterating, Ignore Limiters below** option.

Power Flow Solution Options

This button brings up the [Power Flow Solution Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases and the options used when no contingency-specific solution options are defined. Changing these options affects all power flow computations, even those outside the ATC process.

Because the goal is to stress the system while performing the ATC analysis, there are two power flow solution options that are set internally as part of the ATC process. These are the options that specify the minimum per unit voltage for constant power and constant current loads. These options are both set to 0 pu and override any user-specified settings.



Minimum Per Unit Voltage for	
Constant Power Loads	0.000
Constant Current Loads	0.000

These options are both set to 0 pu during both the base case solution and any contingency case solution.

Injection Group Options

This button brings up a dialog from which options associated with injection groups can be set. These are the same options that get set for injection groups when using [Island-based AGC](#) and dispatching using an injection group; the ATC process switches to using an island-based AGC when iterating and using injection groups for the seller and buyer. The options that can be set are Island AGC Tolerance, Allow only AGC units to vary, Enforce unit MW limits, Do not allow negative loads, and how reactive power load should change as real power load is changed.

The sink injection group is set as the injection group to use in the dispatch. When implementing the transfer ramping, the source injection group is ramped to the required amount and then the power flow is solved with the sink injection group picking up any required changes. These changes include adjusting injection in response to change in the source injection group as well as any changes in losses due to the transfer.

Define Contingency Solution Options

Click this button to open the [Contingency Solution Options Dialog](#). These are the same options that are set with the contingency analysis options.

Use Specific Solution Options

When checked, the ATC tool will use the solution options defined by pressing the **Define Contingency Solution Options** button for contingency analysis. When not checked, all solutions will use the options defined by pressing the **Power Flow Solution Options** button.

Regardless of the specific contingency solution option settings, there are two options that will be set internally by the ATC process and will override any user-specified settings. These are the options for minimum per unit voltage for constant power and constant current loads. These options are described in more detail in the Power Flow Solution Options section above.

Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions

When using the iterated solution methods, the typical iterated operation is to solve for the transfer step amount in the post-contingency state, and then apply the transfer step amount to the post-contingency state and resolve for the next transfer step amount. When this option is checked, the transfer step amount for each iteration is calculated in the post-contingency state, but the power flow state is restored to the pre-contingency state before the transfer step amount is applied. The contingency is then applied after the transfer step amount has been ramped and the transfer step amount for the next iteration is calculated after the contingency has been applied. The purpose of this is to take into account the possibility of conditional actions in the contingency definitions that may not be active in the early steps of the iterated calculations, but may become active at some point during the application of the iterated steps.

Iterate on failed contingency

This option is only enabled when the option to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions** is selected. When this option is checked and the power flow fails to solve after the contingency has been applied, a new iteration loop is

entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current step size. To start the iterative process, the current step size is reduced by half. The ramping is performed and the contingency is solved. The step size will remain the same as long as the contingency can be solved. If the contingency fails to solve, the step size will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original step size, or the step size becomes smaller than the step size tolerance.

An additional check is done during the iterative process on the failed contingency. When the contingency does solve, the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the step size will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved.

During the entire iterative process on a failed contingency solution, the step size is limited to be between zero and the step size which was ramped when the contingency failed to solve for the first time.

Tolerances in ATC Tool

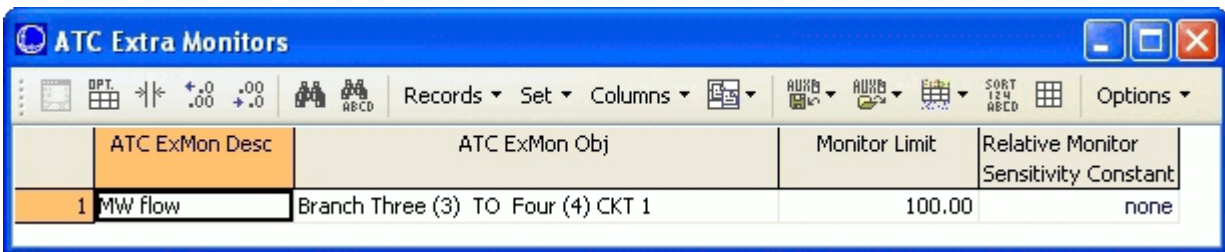
When using one of the iterated methods, tolerances must be set correctly with the ATC tool or injection changes will end up being picked up by the system slack instead of the seller and buyer or no injection changes will be made at all. When studying a transfer between injection groups, the **Transfer Tolerance** will dictate what the **MVA Convergence Tolerance**, set with the [Power Flow Solution Options](#) on the [Simulator Options Dialog](#), and the **Island-Based AGC Tolerance** can be. Before the analysis starts, the MVA Convergence Tolerance will be checked to make sure that it is less than $0.1 * (\text{Transfer Tolerance})$. If not, the MVA Convergence Tolerance will be set to $0.1 * (\text{Transfer Tolerance})$. The Island-Based AGC Tolerance will also be checked to make sure that it is less than $0.5 * (\text{Transfer Tolerance})$. If not, the Island-Based AGC Tolerance will be set to $0.5 * (\text{Transfer Tolerance})$. The MVA Convergence Tolerance will then be checked to make sure that it is less than $0.2 * (\text{Island-Based AGC Tolerance})$. If not, the MVA Convergence Tolerance will be set to $0.2 * (\text{Island-Based AGC Tolerance})$. This will order the tolerances such that $(\text{MVA Convergence Tolerance}) < (\text{Island-Based AGC Tolerance}) < (\text{Transfer Tolerance})$. The original tolerances will be restored when the initial state stored with the ATC tool is restored.

When studying a transfer between areas or super areas, there are two AGC tolerances to deal with. These are both checked to make sure that they are less than $0.5 * (\text{Transfer Tolerance})$ and are set to that value if they are not. Both of these tolerances should also be greater than the MVA convergence tolerance. The smaller of the two tolerances is multiplied by 0.2 and the MVA convergence tolerance must be less than this value. If not, the MVA convergence tolerance is set to $0.2 * (\text{smaller of the two area/super area AGC tolerances})$. All tolerances will be restored to their original values when the initial state stored with the ATC tool is restored.

ATC Extra Monitors Dialog

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize ATC Extra Monitors. To open the Extra Monitors dialog, click the **Define Extra Monitors** button on the [Advanced Options](#) tab of the ATC Analysis Dialog.

The ATC Extra Monitors Display lists all of the ATC Extra Monitors defined. This list display is a [Case Information Display](#), and therefore has the same functionality as other common displays. See **Adding and Removing Extra Monitors** below.



	ATC ExMon Desc	ATC ExMon Obj	Monitor Limit	Relative Monitor Sensitivity Constant
1	MW flow	Branch Three (3) TO Four (4) CKT 1	100.00	none

Extra Monitors Dialog

The default fields shown on this display are:

ATC ExMon Desc

This is a description of the monitored value. Presently, this is always MW flow.

ATC ExMon Obj

The power system element that is being monitored. This will be either a transmission branch or an interface.

Monitor Limit

This is the MW limit of the element being monitored.

Relative Monitor

Set this to a positive value to further filter the Transfer Limitations reported on the [Transfer Limiters Display](#). By default, Relative Monitor is set to *none*, and no additional filtering of limitations is performed. If this value is greater than zero, only Transfer Limitations that meet the following condition are included in the results.

$$\frac{\left(\frac{OTDF_{M,C}}{LimitUsed_M} \right)}{\left(\frac{PTDF_{ExtraMon}}{Rating_{ExtraMon}} \right)} \geq (Relative\ Monitor\ Sensitivity\ Constant)$$

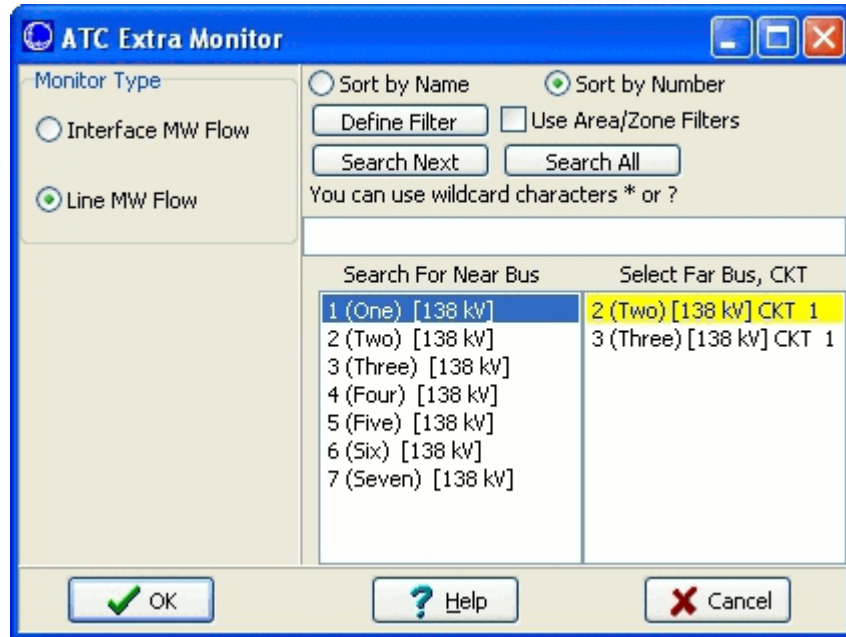
This provides a measure of how much an interface or branch is affected by the transfer relative to its MW limit.

The relative monitor sensitivity check is not done if using one of the [iterated ATC methods](#) and in the process of iterating on individual limiters. The assumption is that once a limiter meets the initial check and it is being iterated on that the limiter should be kept regardless of its relative monitor sensitivity.

Adding and Removing Extra Monitors

To delete an Extra Monitor, right-click on the desired record on the list display and select **Delete**.

To insert an Extra Monitor, right-click on the list display and select **Insert**. This opens the insert dialog.



Insert Extra Monitors Dialog

Choose whether you want to monitor an Interface or Line MW flow. Next choose the interface or branch to be monitored. Note that the insert dialog allows the use of Simulator's [Advanced Search Engine](#) and techniques to aid in locating the desired interface or branch. Click **OK** to insert the record.

ATC Dialog: Result

The Result page is found on the [Available Transfer Capability dialog](#). This page is visible when performing ATC analysis on a single system state. The Result page consists of two sections: A [Transfer Limiters Display](#) and a [Contingency Definition Display](#).

The screenshot shows the 'Available Transfer Capability' dialog box with the 'Result' tab selected. The left sidebar contains a tree view with 'Options', 'Analysis', and 'Result' (selected). The main area displays a table of transfer limiters under the 'All Limiters' tab. The table has columns: 'Trans Lim', 'Limiting Element', 'Limiting CTG', '% OTDF', and 'Pre-Est'. Below the table is a 'Contingency Definition' section with a table showing actions for a specific contingency.

	Trans Lim	Limiting Element	Limiting CTG	% OTDF	Pre-Est
1	-110.56	Branch One (1) TO Three (3)	L_000001One-000002TwoC1	33.33	10
2	-11.98	Branch Two (2) TO Five (5)	CL_000002Two-000006SixC1	65.71	10
3	70.10	Branch Two (2) TO Five (5)	CL_000004Four-000005FiveC1	16.67	8
4	73.18	Branch Four (4) TO Five (5)	CL_000002Two-000005FiveC1	16.21	4
5	101.19	Branch Four (4) TO Five (5)	CL_000002Two-000006SixC1	34.29	2
6	137.20	Branch Two (2) TO Six (6)	CKL_000002Two-000005FiveC1	83.79	8
7	144.44	Branch One (1) TO Two (2)	CL_000001One-000003ThreeC1	33.33	10
8	176.51	Branch Four (4) TO Five (5)	CL_000001One-000002TwoC1	19.19	2
9	186.17	Branch Two (2) TO Six (6)	CKL_000004Four-000005FiveC1	83.33	4
10	187.57	Branch Two (2) TO Five (5)	CL_000001One-000003ThreeC1	8.30	8
11	200.00	Branch Two (2) TO Six (6)	CKL_000007Seven-000005FiveC1	100.00	
12	200.36	Branch Two (2) TO Six (6)	CKL_000002Two-000003ThreeC1	78.30	4
13	200.99	Branch Two (2) TO Six (6)	CKL_000002Two-000004FourC1	77.93	4

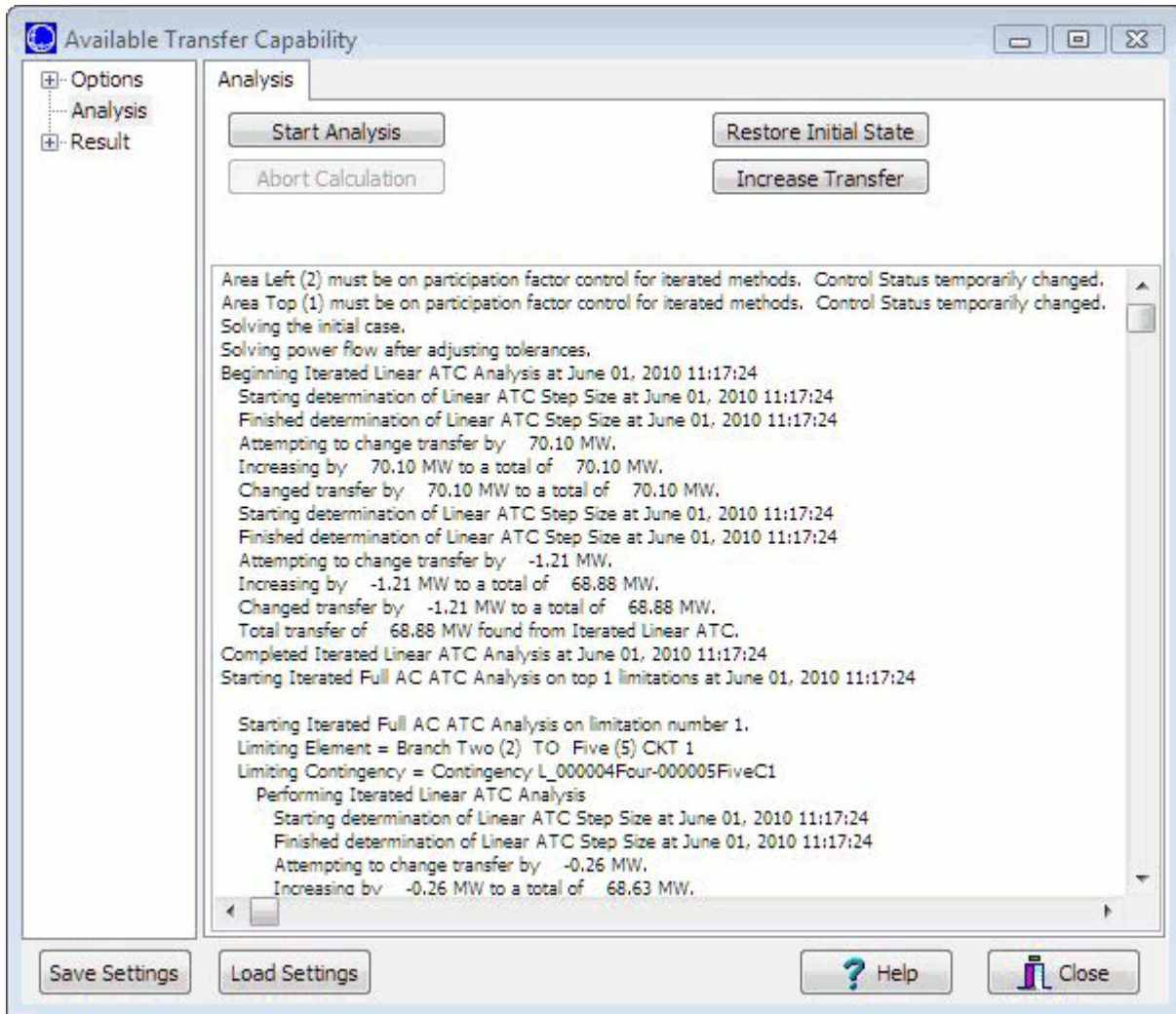
Actions	Model Crite
1 OPEN Line One_138.0 (1) TO Two_138.0 (2) CKT 1	

The Transfer Limiters Display contains tabbed pages containing information on Branch, [Interface](#), and [Nomogram](#) Limiters. The user can also choose to display All Limiters. See [Transfer Limiters](#) for more information.

The Contingency Definition section displays information on the limiting contingency for selected transfer limiter if the user checked **Include Contingencies** on the [Common Options](#) tab.

ATC Dialog: Analysis

The Analysis page, found on the [Available Transfer Capability dialog](#), is used to control the analysis process. The scrollable window displays a record of user-initiated actions relating to the analysis.



The Analysis page has the following controls:

Start Analysis

The **Start Analysis** button begins the ATC Analysis using the settings and options defined by the user in the ATC Analysis Dialog. Progress of the ATC Analysis is shown in the scrollable pane of the ATC dialog – Analysis page window. Note that you can still interact with other features of Simulator when the ATC dialog is open, and care must be taken while actually running the ATC solution, as any changes to the data while the ATC solution is calculating will invalidate the results.

Abort Calculation

When using one of the iterative ATC Solution Methods, or while analyzing Multiple Scenarios, click this button to abort the calculation.

Note: This does not immediately abort the solution. Simulator must restore the system state before completing the abort. Also, after aborting the solution, it should be assumed that the results are not valid. The exception could be if multiple scenarios have fully completed before hitting the abort button. Any study or scenario in progress when Abort is clicked should definitely be considered not valid.

Restore Initial State

Click this button to restore the system state to the initial state. The initial state is the state in memory when the ATC analysis is started. The initial state is only stored if using one of the iterative ATC Solution Methods or analyzing Multiple Scenarios. The initial state is also stored when opening the ATC dialog if an initial state has already been stored.

Increase Transfer

Click this button to open the Ramp Transfer Up dialog in order to increase (or decrease) the transfer level manually. This transfer remains in the system state if the dialog is closed without restoring the initial state.

Additional analysis can be done on a case that has had the transfer increased by using the ATC tool; however, the control method used in the ATC tool may differ from that in use once the ATC tool has been exited. When injection groups are used to increase the transfer, the AGC method that is used is island-based AGC. The AGC method is then restored to what it was prior to increasing the transfer. If this same method is not used for subsequent analysis, the dispatch and system state will be modified from the state with the transfer applied. When using areas or super areas for increasing the transfer, the area or super area is put on participation factor control for the ramping. Areas/super areas participating in the transfer will remain on participation factor control following the ramping, but the same problem can exist as with the injection groups if the same AGC method is not in use with subsequent analysis after the transfer ramping.

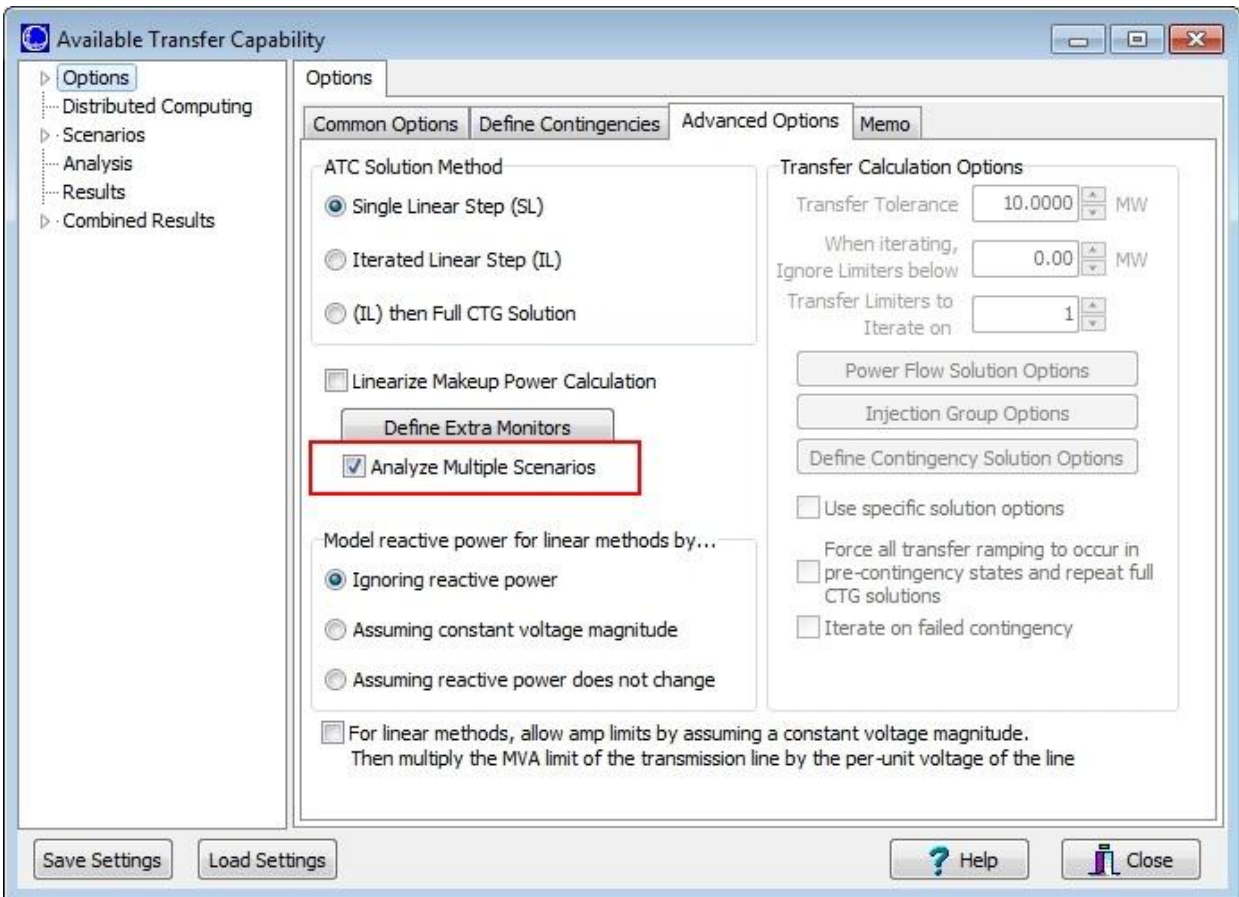
If using areas or super areas for increasing the transfer, a transaction is added to the [MW Transactions](#) to implement the transfer. This transaction can be identified by the special ID that is assigned to it, *TEMP_ATC_TRANSACTION*. This transaction will remain in the case if closing the ATC dialog without restoring the initial state.

When increasing the transfer, the rules for the various tolerances involved in the transfer and power flow solution described in the **Tolerances in ATC Tool** section described in the [Advanced Options](#) topic apply. The only exception is that the original tolerances are not restored at the end of the transfer increase because the initial state is not restored.

Multiple Scenario Available Transfer Capability Dialog

To perform [Available Transfer Capability Analysis](#) for several system scenarios, check **Analyze Multiple Scenarios** on the **Advanced Options** tab of the [Available Transfer Capability dialog](#). When Analyze Multiple Scenarios is not checked, the available pages are [Options](#), [Analysis](#) and [Result](#). When Analyze Multiple Scenarios is checked then the following changes occur:

- [Distributed Computing](#) page appears
- [Scenarios](#) page appears
- [Results](#) page appears
- Result page is removed (replaced by Results)
- [Combined Results](#) page appears



By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

- Line Rating/Zone Load Scenarios (weather-related scenarios)
- Generation Scenarios (generation profiles)
- Interface constraints

See [Scenarios](#) page for more information.

Available Transfer Capability : Distributed Computing

The Distributed Contingency Analysis tool is available as an add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for more details.

These options are all available on the [Available Transfer Capability Dialog](#) under the Distributed Computing Grouping.

Distributed Computing is available for use in distributing [multiple scenario ATC simulations](#). In order to use distributed computing you must first configure a list of remote computers which can be utilized along with appropriate authentication information for those computers. The computer list and authentication information is common to all the distributed computing tools in Simulator and can be found in the [Simulator Options Dialog](#), or reached with the Distributed Computing Options button. They are described in [Distributed Computing Options](#).

The only option specific to ATC Analysis for Distributed Computing is the following

Use Distributed Computing

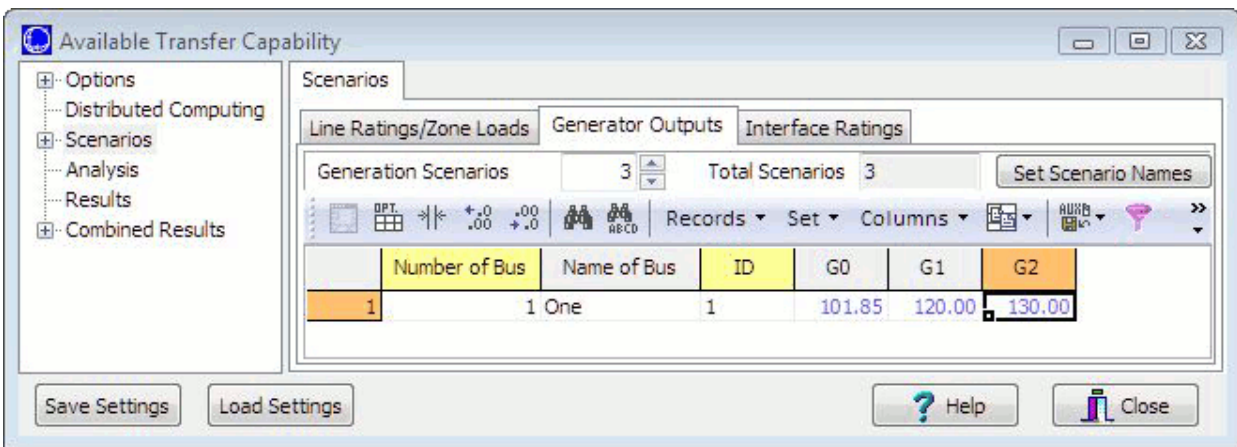
Check this box to signify that when processing [multiple scenario ATC simulations](#) distributed computing should be used.

Multiple Scenario ATC Dialog: Scenarios

The Scenarios page of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability Dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability dialog](#).

By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

- Line Rating/Zone Load Scenarios (weather-related scenarios)
- Generation Scenarios (generation profiles)
- Interface constraints



Each of the different scenario types can be defined on its own tab. Each tab contains a list of the power system elements that will be modified during different scenarios. These lists are a familiar [Case Information Display](#) providing the same functionality as other displays.

To insert a new power system element into the list, right-click on the list (below the headings) and choose **Insert**.

If a power system element does not show up in one of the lists of elements for a particular scenario, then it will retain its base case value during that scenario.

When saving or loading scenario elements in an auxiliary file, the elements are usually identified by primary key fields. Optionally, these elements can be identified by label. See the [Label](#) topic for more information about this.

Line Ratings/Zone Loads Tab

Use this tab to define Line Ratings and Zone Loads scenarios.

Line Ratings and Zone Loads are varied together when analyzing scenarios. This was chosen because they often vary together as a function of the weather. Notice that three sub-tabs (labeled Line Ratings A, Line Ratings B, and Zone Loads) appear at the top of the Line Ratings/Zone Loads tab.

Line Ratings A and B

The A rating is typically used as a base case rating, whereas the B limit is typically used as a line rating under contingency. The way line ratings are utilized is defined in [Limit Monitoring Settings](#).

A checkbox, **Only Monitor Scenarios Limits**, appears on both of these sub-tabs. Check this box to override the normal Limit Monitoring Settings and monitor only the lines that are defined as part of either the Line Ratings A or Line Ratings B lists. When this option is in use, rating A will be used for base case limitations, and rating B will be used for contingency limitations. If a line is not in both lists, the rating that is specified will be used for both base case and contingency limitations. During the ATC calculations, Limit Monitoring Settings will be modified internally to only monitor the specified lines. Once the analysis is complete, the Limit Monitoring Settings will be their original settings.

Zone Loads

The Zones sub-tab has additional options used to specify how Mvar load should vary as the MW load is updated. Only MW load is specified in the scenarios. To specify how the Mvar load should vary, select either **Assume Constant Power Factor** to have the Mvar load vary with the MW by the same power factor that exists in the base case or **No Change in Mvar** to keep the Mvar load constant.

When determining how zone loads should be modified during each scenario, the AGC status of each load is used to determine how the MW portion of the load should be modified. For each zone to be modified, the Total MW load for all connected loads in the zone is determined regardless of AGC status. The Total AGC MW load for all connected loads in the zone is determined for only those loads where AGC = YES. If Total AGC MW > 0 then only those loads where AGC = YES will be modified. The multiplier for load changes becomes, $\text{multiplier} = 1 + (\text{Load MW Change})/(\text{Total AGC MW})$. If Total AGC MW = 0, all loads in the zone will be modified during the scenario regardless of the AGC status and the multiplier becomes $\text{multiplier} = 1 + (\text{Load MW Change})/(\text{Total MW})$. Load MW Change is determined by taking the new MW load value specified with the scenario and subtracting off the Total MW, $(\text{Load MW Change}) = (\text{New MW Load Value}) - (\text{Total MW})$. If Total MW = 0 meaning that there is presently no connected load in the zone, no load changes will be made. If Total AGC MW > 0, only those loads where AGC = YES that are in-service will be modified with the new MW load at each load becoming the present MW load multiplied by the multiplier $(1 + (\text{Load MW Change})/(\text{Total AGC MW}))$. If Total AGC MW = 0, all loads that are in-service will be modified with the new MW load at each bus becoming the present MW load multiplied by the multiplier $(1 + (\text{Load MW Change})/(\text{Total MW}))$.

Generator Outputs Tab

Use this tab to define scenario changes for generator MW output. Specify a zero output to take a generator off-line. Specifying a non-zero output will force the status of the generator to be closed, meaning that a generator that is currently off-line can be brought on-line during a scenario. The AGC status of a scenario generator will be set to NO so that any changes made via a scenario will not be undone by any automatic generation control schemes. If the bus to which the generator is connected is not in-service, simply changing the generator status to closed will not put the generator in-service. If the case has breakers defined, an attempt will be made to ensure that a generator is truly in-service by closing any open breakers that would cause it to be out-of-service. This is done whenever breakers have been defined in the case. Transmission branches can be identified as breakers by using the **Branch Device Type** field on the [Line and Transformers Display](#).

Interface Ratings Tab

Use this tab to define scenario changes for interface MW ratings.

Applying Scenarios

When scenarios are applied, all of the changes specified in the scenario are not applied at once. Generator and load changes that cause injection changes will force the power flow to solve if the

injection change exceeds a certain threshold. Currently this threshold is 500 MW. Changes will accumulate until this condition is met and then the power flow will be solved. If the power flow fails to converge at any point during this process, the process for making changes for that type of scenario change is aborted, e.g. if there are 10 load changes to be made and accumulated change for the first 5 changes exceeds 500 MW the power flow is solved. If it fails to converge, the changes for the last 5 load changes will not be applied. The system state that remains will not be a state that contains all of the changes for the scenario. Over all scenario types changes are applied in the following order: Line Rating A, Line Rating B, Generator, Load, and Interface. The power flow will only be applied for the changes that affect injection, i.e. generators and loads. If the power flow fails at any step during this process, the remaining changes will not be applied.

The Scenarios page has these additional controls:

Number of Defined Scenarios per Element Type

On each tab, you may enter how many different scenarios should be defined for that kind of power system element. For instance if you set Generation Scenarios to 5, then list display on the Generation Tab will provide 5 columns labeled G0, G1, G2, G3, and G4. Generation outputs should then be entered into each cell representing the generation output in each scenario.

Total Scenarios

Once you have specified the scenarios, Simulator is able to perform ATC Analysis on every combination of the axes. For example, assume you have the following:

- 10 sets of line ratings and zones load
- 8 sets of generation profiles
- 3 interface constraints

This yields a total of 240 different scenarios to calculate ($10 \times 8 \times 3 = 240$). Be warned that the more scenarios you analyze the longer the computation will take.

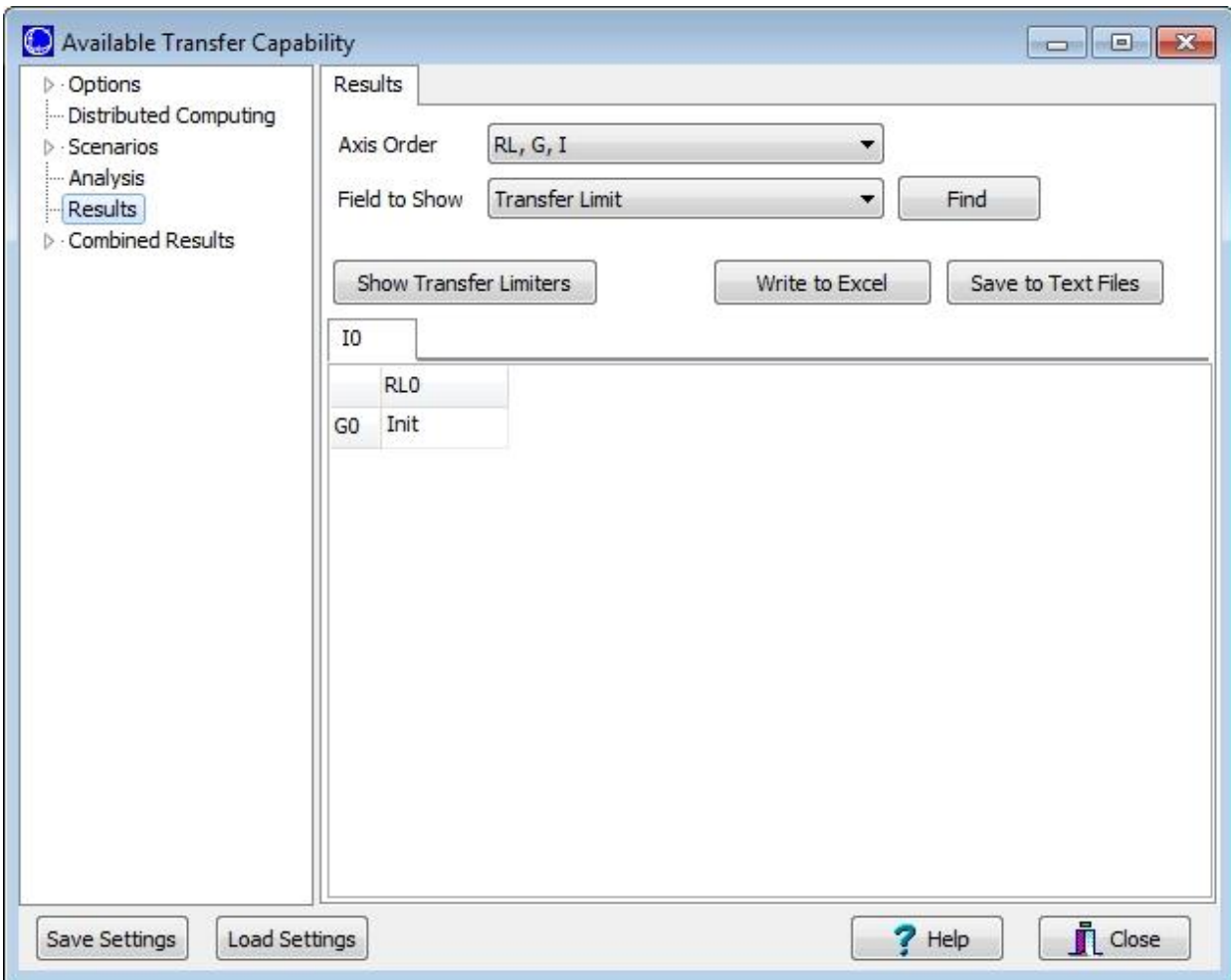
Set Scenario Names

This button will bring up the Scenario Names dialog, where a different name can be assigned to each scenario.

Multiple Scenario ATC Dialog: Results

The Results page of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability dialog](#).

Once you have defined your scenarios and started the ATC Analysis, you can switch to the Results page to see the progress that ATC Analysis is making.



The primary part of the Results page contains a spreadsheet-like look-up table display. The layout of the display is dictated by the **Axis Order** selected. Note that because there are three axes possible, the row and column entries represent the first two axes while the third axis is represented by tabs at the top of the table. See [Local Menu Options](#) for information regarding the options available when right-clicking within the table.

This dialog has the following controls:

Axis Order

This menu is used to select the desired axis order. The three axes correspond to the three [Scenarios tabs](#).

- One axis has heading labels G0, G1, ... for the Generation Scenarios,
- Another with heading labels RL0, RL1, ... for the Rating/Load Scenarios,
- A third with heading labels I0, I1, ... for the Interface Scenarios.

Field to Show

Use this dropdown to select which field is shown in the **Results Display**. By default this is set to Transfer Limit, but any field associated with a transfer limiter can be displayed. Click the **Find** button to open a dialog that allows search options for determining which field to select.

Show Transfer Limiters

Click the **Show Transfer Limiters** button to view the [Transfer Limiters](#) found under each scenario. This will open a separate dialog that displays a list of the Transfer Limiters. To see the Transfer Limiters for a particular scenario, click on the workbook cell that represents the scenario you are interested in and the separate dialog will update appropriately.

Write to Excel

This button will send the results to an Excel spreadsheet.

Save to Text Files

This option will allow the user to save the results in an auxiliary file.

Results Display

The primary part of the Results tab contains a spreadsheet-like look-up table display. The values shown in the table for each scenario are for the field selected with the **Field to Show** option. By default this is the Transfer Limit. The value that is reported corresponds to the transfer limiter with the most limiting Transfer Limit. If using one of the iterated methods, the most limiting Transfer Limit is determined from all limiters that have been iterated on.

Multiple Scenario ATC Analysis Results: Local Menu Options

The [Results](#) page of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability dialog](#).

Right-clicking on the table contained on the Results page will bring up a local menu containing several other options. The first several options are only enabled if a cell that represents a scenario is right-clicked. These options will perform an operation with respect to the Scenario:

Take me to Scenario ...

Implements the changes in the Scenario resulting in a modified system state. Transfer limits are not calculated.

Determine Transfer Limit For Scenario ...

Calculates the ATC for the Scenario, and then sets the system state back to the Initial State.

Determine Transfer Limit for Selected Scenarios

This option is available if multiple scenarios are selected. Multiple scenarios can be selected by holding down the CTRL key and left-clicking on the desired scenarios. When this option is selected, transfer limits for all of the highlighted scenarios will be calculated. The system state will be restored to the Initial State upon completion of all calculations.

Take me to the Transfer Limit For Scenario ...

Performs ATC analysis for the Scenario, and then ramps the transfer to the lowest transfer limit found. If using one of the iterated methods, the transfer limit used for the ramping is the lowest one that is iteratively found.

Other options on this local menu are not related to the Scenario that has been clicked.

Increase Transfer for Present System State

Increments the transfer level for the present system state by a user-defined amount.

Return to Initial State

Returns the system state to the Initial State. The initial state is the state in memory when the ATC analysis is started. The initial state is only stored if using one of the iterative ATC Solution Methods or analyzing Multiple Scenarios. The initial state is also stored when opening the ATC dialog if an initial state has already been stored.

Multiple Scenario ATC Dialog: Combined Results

The Combined Results page of the Multiple Scenario ATC Analysis is available on the [Available Transfer Capability dialog](#) when selecting the option to **Analyze Multiple Scenarios** on the [Advanced Options](#) tab of the [Available Transfer Capability dialog](#).

Available Transfer Capability

Combined Results

Options
Distributed Computing
Scenarios
Analysis
Results
Combined Results

Records Set Columns

All Limiters Branch Limiters Interface Limiters Nomogram Interface Limiters

	ATCSce Line/Zone Scenario Number	ATCSce Gen Scenario Number	ATCSce Interface Scenario Number	Trans Lim	Limiting Element	Limiting CTG
18	0	0	0	396.33	Branch Two (2) TO Four (4) CL_000003Three-000004FourC	
19	0	0	0	582.51	Branch Two (2) TO Three (3) L_000001One-000002TwoC1	
20	0	0	0	1194.39	Branch One (1) TO Two (2) CL_000006Six-000007SevenC1	
21	0	0	0	1445.39	Branch Six (6) TO Seven (7) CL_000006Six-000007SevenC2	
22	0	1	0	-18.50	Branch Two (2) TO Five (5) CL_000002Two-000006SixC1	
23	0	1	0	66.69	Branch Two (2) TO Five (5) CL_000004Four-000005FiveC1	
24	0	1	0	79.24	Branch Four (4) TO Five (5) CL_000002Two-000005FiveC1	
25	0	1	0	104.09	Branch Four (4) TO Five (5) CL_000002Two-000006SixC1	
26	0	1	0	136.22	Branch Two (2) TO Six (6) CKL_000002Two-000005FiveC1	
27	0	1	0	183.68	Branch Four (4) TO Five (5) CL_000001One-000002TwoC1	
28	0	1	0	185.93	Branch Two (2) TO Six (6) CKL_000004Four-000005FiveC1	
29	0	1	0	193.76	Branch Two (2) TO Five (5) CL_000001One-000003ThreeC1	

Contingency Definition

	Actions	Model Criteria	S
1	OPEN Line Two_138.0 (2) TO Six_138.0 (6) CKT 1		

Save Settings Load Settings Help Close

The tables on the Combined Results page show the transfer limitations under ALL scenarios. The scenarios are identified by the default fields ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges. When defining [scenarios](#) for Line Ratings/Zone Loads, Generator Outputs, and Interface Ratings each corresponding scenario is given an integer identifier. The entries in the table fields correspond to these integer identifiers. Transfer limitations can be filtered based on the type of limitation by switching between the All Limiters, Branch Limiters, Interface Limiters, and Nomogram Interface Limiters sub-tabs, but the tables on any of these sub-tabs will still contain the relevant type of limiters from all scenarios. The transfer limiter results and remaining fields correspond to those described in the [Transfer Limiters Display](#) section.

The ATCLineZoneChanges, ATCGenChanges, and ATCInterfaceChanges fields can be used when interacting with the ATC results using auxiliary files and SimAuto to get access to the multiple scenario results. These fields should be used with the TRANSFERLIMITER data type to specify the ATC scenario to which a transfer limitation corresponds. These fields are extra key fields for this data type. If these fields are blank or omitted from input data, the limitations are assumed to go with the standard single set of ATC results.

ATC Dialog: Transfer Limiters Display

The Transfer Limiters Display shows the results of [ATC Analysis](#). This display appears either on the [Result page](#) of the [Available Transfer Capability dialog](#) or as a separate window when performing [Multiple Scenario Available Transfer Capability Analysis](#).

The display contains four different tabs: All Limiters, Branch Limiters, Interface Limiters, and Nomogram Interface Limiters.

The Contingency Definition display at the bottom of the window shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

The screenshot shows the 'Available Transfer Capability' dialog box. The 'Result' tab is selected in the sidebar. The main area displays a table of transfer limitations under the 'All Limiters' tab. The table has columns: Trans Lim, Limiting Element, Limiting CTG, % OTDF, Pre-Trans Est, and Limit Used. Below the table is a 'Contingency Definition' section with a table showing the selected contingency.

	Trans Lim	Limiting Element	Limiting CTG	% OTDF	Pre-Trans Est	Limit Used
1	-110.56	Branch One (1) TO Three (3)	L_000001One-000002TwoC1	33.33	101.85	65.00
2	-11.98	Branch Two (2) TO Five (5)	CL_000002Two-000006SixC1	65.71	107.87	100.00
3	70.10	Branch Two (2) TO Five (5)	CL_000004Four-000005FiveC1	16.67	88.32	100.00
4	73.18	Branch Four (4) TO Five (5)	CL_000002Two-000005FiveC1	16.21	48.14	60.00
5	101.19	Branch Four (4) TO Five (5)	CL_000002Two-000006SixC1	34.29	25.30	60.00
6	137.20	Branch Two (2) TO Six (6)	CK_L_000002Two-000005FiveC1	83.79	85.04	200.00
7	144.44	Branch One (1) TO Two (2)	CL_000001One-000003ThreeC1	33.33	101.85	150.00
8	176.51	Branch Four (4) TO Five (5)	CL_000001One-000002TwoC1	19.19	26.13	60.00
9	186.17	Branch Two (2) TO Six (6)	CK_L_000004Four-000005FiveC1	83.33	44.86	200.00
10	187.57	Branch Two (2) TO Five (5)	CL_000001One-000003ThreeC1	8.30	84.43	100.00
11	200.00	Branch Two (2) TO Six (6)	CK_L_000007Seven-000005FiveC1	100.00	0.00	200.00
12	200.36	Branch Two (2) TO Six (6)	CK_L_000002Two-000003ThreeC1	78.30	43.11	200.00
13	200.99	Branch Two (2) TO Six (6)	CK_L_000002Two-000004FourC1	77.93	43.37	200.00
14	213.23	Branch Four (4) TO Five (5)	CL_000003Three-000004FourC1	16.00	25.88	60.00

	Actions	Model Criteria	Status
1	OPEN Line One_138.0 (1) TO Two_138.0 (2) CKT 1		CHECK

Transfer Limiters Display

The Transfer Limiters Display has the following controls:

All Limiters Tab

The All Limiters tab shows a list of all the transfer limitations found. This includes limitations on branches, interfaces, areas, zones, etc...

Branch Limiters Tab

The Branch Limiters tab only shows those limitations with a transmission line or transformer as the limiting element. The limiting element description for the Branch Limiters tab is replaced by the bus numbers and names of the limiting element.

Interface Limiters Tab

The Interface Limiters tab only shows those limitations with an interface as the limiting element. The limiting element description is replaced by the name of the interface.

Nomogram Interface Limiters Tab

The Nomogram Interface Limiters tab only shows those limitations with a nomogram interface as the limiting element. The limiting element description is replaced by the name and segment of the nomogram interface.

Limiters Fields

Transfer Limit (shown by default)

The Transfer Limit Field shows the Transfer Limit in MW for the Limiting Element during the Limiting Contingency. This value depends on the ATC Solution Method Used:

- **Single Linear Step (SL)**: Only one Linear ATC step is performed. The Transfer Limit values are those found during this step.
- **Iterated Linear Step (IL)**: The Single Linear Step method is iterated during this method. The Transfer Limiters shown are those found during the final step performed. The actual Transfer Limit values are the values found at the last step plus the accumulated amount the transfer has been ramped. The Transfer Limitation(s) that were iterated on individually are highlighted in yellow.
- **Iterated Linear Step (IL) then Full CTG**: The Transfer Limits shown for the Transfer Limitation(s) that were used when iterating are equal to the accumulated amount that the transfer was ramped during the initial Iterated Linear Step and then the contingency analysis step. These limitations are highlighted in cyan. The Transfer Limits shown for all other limiters are the results of the initial Iterated Linear Step calculation performed before iterating on individual limitations for the contingency analysis.

For the iterated methods the Iteratively Found field, Limiting Element field, and color indicators are used to highlight the Transfer Limitations when certain situations are encountered. Unless otherwise indicated, the keyword listed below will show up in the Iteratively Found field:

- **YES (yellow)** - Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) method or Iterated Linear Step (IL) then Full CTG method when studying base case limitations. The Transfer Limit that is reported is the thermal limitation.
- **FULL (aqua)** - Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) then Full CTG method. The Transfer Limit that is reported is the thermal limitation.
- **OSCILLATING (lime green)** - This keyword will show up in the Limiting Element field. This indicates that the number of iterations has exceeded the maximum number of allowed iterations. This is set at 100 internally. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. It is very unlikely that this limitation should occur because there are other mechanisms in place to prevent oscillations in the process.
- **POWERFLOW_DIVERGENCE (fuchsia)** - If the Limiting Element field also contains *POWERFLOW_DIVERGENCE* this indicates that this type of limit was encountered when iterating on all limiters at the start of the process. If the Limiting Element lists an actual limiting element, this indicates that this type of limit was encountered during the process

where an individual limiter is being iterated on. If using the Iterated Linear Step (IL) then Full CTG method this is the step that occurs before actually applying the contingency. The Transfer Limit that is reported is the total amount that could be ramped before the power flow failed to converge. This transfer limitation means that the full desired transfer amount cannot be achieved.

- **RAMP_FAIL_IN_FULL (fuchsia)** - If using the Iterated Linear Step (IL) then Full CTG method and forcing the transfer ramping to occur pre-contingency, this type of limiter will occur only when iterating on individual limiters. This indicates that the power flow was not able to solve while ramping the transfer and not because the contingency failed to solve. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution at which the contingency solved.
- **RAMP_FAIL_IN_BASE (gray)** - This type of limiter will not be reported for new studies run with Simulator version 16 or later. It is being retained for the purposes of reporting limiters that were saved with Simulator version 15 and earlier. This type of limitation is the same as a *POWERFLOW_DIVERGENCE* limiter and would only be reported for a transfer limitation for the base case or when using the Iterated Linear Step (IL) method.
- **RAMP_FAIL_IN_FULL_AFTER_CTG (gray)** - This type of limiter would only be reported if using the Iterated Linear Step (IL) then Full CTG method and ramping the transfer post-contingency when the full ramping amount cannot be achieved. The Transfer Limit that is reported is the total amount that could be ramped before the power flow failed to converge.
- **CTG_FAIL_IN_FULL (orange)** - Indicates that the contingency failed to solve at some point during the Iterated Linear Step (IL) then Full CTG method. This type of limiter would be reported when NOT using the option to iterate on failed contingencies. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution in which the contingency solved. If the contingency fails to solve immediately following the completion of the iterated process on the individual limiter without the contingency implemented, the Transfer Limit that is reported is the accumulated transfer amount achieved from that process without the contingency implemented.
- **CTG_FAIL_ITERATED (red)** - Indicates that the contingency failed to solve at some point during the Iterated Linear Step (IL) then Full CTG method and using the option to iterate on failed contingencies. The Transfer Limit that is reported is the accumulated amount that is possible without overloading the monitored element and in which the contingency solves. Using the iterated method attempts to get as close as possible (within specified tolerances) to the highest transfer amount at which the contingency will solve. This is very similar to a PV nose point.
- **CTG_FAIL_IN_BASE (purple)** - Indicates that the contingency failed to solve at zero transfer level (base case failure) when using the Iterated Linear Step (IL) then Full CTG method and choosing to iterate on failed contingencies. The Transfer Limit that is reported is the accumulated transfer amount that was ramped during the iterated process on the individual limiter without the contingency implemented.

Limiting Element (shown by default)

Shows a text description of the limiting element. This will contain either the actual limiting element or some additional information about a limiter. Special keywords that can appear are *OSCILLATING* and *POWERFLOW_DIVERGENCE*. There are described in more detail above in the **Transfer Limit** description. If reporting reserve limits and either the source or sink will hit a reserve limit at a particular transfer level, the name of the source or sink will appear as the limiting element.

Limiting CTG (shown by default)

Shows the name of the limiting contingency.

% OTDF (shown by default)

This is the OTDF (or PTDF if the Limiting CTG is *Base Case*) on the Limiting Element for the transfer direction that is being studied. In other words, this is a linear estimate of the percent of the transfer that will appear on the Limiting Element if the Limiting CTG occurs.

Note: For iterated techniques, this is the PTDF or OTDF at the last Linear Iteration.

Pre-Transfer Value Estimate (shown by default)

If a contingency is not included in the Limiter, this is equal to the Initial Value. When a contingency is included in the Limiter, this is the linear estimate of the post-contingency flow before any transfer occurs. See [Available Transfer Capability Analysis](#).

Note: For iterated techniques, this is the estimate at the last Linear Iteration.

Limit Used (shown by default)

This is the value of the Limit being used by the ATC for the Limiting Element during the Limiting CTG. It reflects what is specified in [Limit Monitoring Settings](#) and options set on the [Advanced Options](#) tab.

MVA the Limit Used is based on

This is the unaltered MVA limit that is defined with a particular limiting element. This is the starting point for determining the **Limit Used** that reflects [Limiting Monitoring Settings](#) and options that are set on the [Advanced Options](#) tab.

Iteratively Found

String indicating if the limitation was found using one of the [iterated methods](#). Possibilities for this are *NO*, *YES*, *FULL*, *CTG_FAIL_IN_FULL*, *CTG_FAIL_IN_BASE*, *CTG_FAIL_ITERATED*, *POWERFLOW_DIVERGENCE*, *RAMP_FAIL_IN_FULL*, and *RAMP_FAIL_IN_FULL_AFTER_CTG*. The various entries correspond to the limit being highlighted in different colors. Additional information about what the different entries mean is given above in the **Transfer Limit** description.

Contingency Definition Display

This display shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

ATC Analysis Methods: Solution Methods

Simulator provides three methods of determining ATC.

- [Single Linear Step \(SL\)](#)
- [Iterated Linear Step \(IL\)](#)
- [Iterated Linear Step \(IL\) then Full Contingency Solution](#)

ATC Analysis Methods: Single Linear Step (SL)

The Single Linear Step approach is the most common ATC method. This method of ATC analysis uses only information about the present system state and sensitivities (mathematical derivatives) about the present system state. These sensitivities are embodied in the [PTDF](#) and [LODF](#) calculations.

Consider a transmission line with a limit of 10 MW, present loading of 5 MW and a PTDF of 10%. The estimated maximum transfer without causing an overload on the line is

$$\text{Transfer Limitation} = (\text{Limit} - \text{Present Loading}) / \text{PTDF} = (10 - 5) / 0.1 = 50 \text{ MW}$$

When including contingency analysis, the [OTDF \(Outage Transfer Distribution Factor\)](#) and linearized estimates of post-contingency flows are used to determine the Transfer Limitation.

$$\text{Transfer Limitation} = (\text{Limit} - \text{Post-Contingency Loading}) / \text{OTDF}$$

If we find the Transfer Limitation for every transmission branch (and interface) during each contingency, then the ATC is equal to the smallest Transfer Limit value.

Contingency elements have an associated [Status](#) that determines under what conditions a particular action will be implemented. When ATC values are calculated using the single linear step method, contingency actions are not actually implemented. Because contingency actions are not actually implemented, *POSTCHECK* actions cannot be applied after other actions have been performed and the power flow has been solved. Instead, *POSTCHECK* actions act as *CHECK* actions during the single linear step method. Whether or not model criteria are met for *CHECK* and *POSTCHECK* contingency actions will be determined at the zero transfer level rather than at the transfer level that determines the limiting transfer.

Note: Simulator also monitors the possibility that a transfer will reduce the flow on a line until the line reaches its limit for flow in the opposite direction.

ATC Analysis Methods: Iterated Linear Step (IL)

The Single Linear Step is an extremely fast method for determining the ATC. However, because it only uses present operating point information, controller changes are not taken into account. The linearization assumes that all controllers are fixed. The Iterated Linear Step (IL) method provides an alternative to the Single Linear Step that allows controller changes, but still performs its analysis in a reasonable amount of time. The (IL) method operates as follows:

- Perform Single Linear Step
- Set the Stepsize = Minimum Transfer Limitation found in Single Linear Step which is greater than the specified **When iterating, Ignore Limiters below** value
- If $[\text{abs}(\text{Stepsize}) \leq \text{Tolerance}]$ or $[\text{Iterations} \geq 100]$ then stop
 - o If the maximum 100 number of iterations is reached, a transfer limitation will be reported highlighted in lime green with the Iteratively Found field set to *OSCILLATING* indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. The maximum number of iterations is a value that is set internally. It is unlikely that this limit will ever be reached because other mechanisms are in place to prevent oscillations.
 - o The iteration count is reset at the beginning of the iteration process.
- Implement transfer by amount of Stepsize and resolve power flow
 - o The power flow is solved according to the options set with the [Power Flow Solution Options](#).
 - o When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
 - o To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:
 - If $\text{Stepsize} > 0$ then $\text{LowBound} = (\text{Accumulated Transfer})$
 - Else If $\text{Stepsize} < 0$ then $\text{HighBound} = (\text{Accumulated Transfer})$
 - If $\text{Stepsize} \geq (\text{HighBound} - \text{Accumulated Transfer}) * 0.7$
 - Then $\text{Stepsize} = (\text{HighBound} - \text{Accumulated Transfer}) * 0.7$
 - Else If $\text{Stepsize} \leq (\text{LowBound} - \text{Accumulated Transfer}) * 0.7$
 - Then $\text{Stepsize} = (\text{LowBound} - \text{Accumulated Transfer}) * 0.7$
 - o During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes by reducing the stepsize by half. The stepsize reduction continues until a stepsize is found at which the power flow will solve. Once this stepsize is found, an attempt will be made to ramp to the original full stepsize using the smaller reduced stepsize. If at any point in this process a stepsize cannot be ramped, an iterative process will be used to reduce the

- stepsize so as much as possible of the full original stepsize can be ramped. An internal tolerance is used to prevent the stepsize from becoming too small.
- o If the power flow fails to converge at this point without being able to implement the full Stepsize transfer, a transfer limitation will be reported highlighted in fuchsia with the Iteratively Found field set to *POWERFLOW_DIVERGENCE*. The Transfer Limit that is reported is the total amount that could be ramped before the power flow failed to converge. This transfer limitation means that the full desired transfer amount cannot be achieved.
 - o If the full transfer amount cannot be achieved in this process when all limiters are being iterated on, the iterative process will continue and iterate on individual limiters.
 - o When all limiters are being iterated on, the Limiting Element will show *POWERFLOW_DIVERGENCE* in addition to the Iteratively Found field showing *POWERFLOW_DIVERGENCE*. This indicates exactly where in the process the ramping could not be achieved.
 - o If the full transfer amount cannot be achieved in this process when individual limiters are being iterated on, the analysis method selected will dictate how the process continues. If using the Iterated Linear Step (IL) method, a Transfer Limit will be reported that is the total amount that could be ramped before the power flow failed to converge and this limiter will be highlighted in fuchsia with the Iteratively Found field set to *POWERFLOW_DIVERGENCE*. If the Iterated Linear Step (IL) method is being used as part of the Full CTG Solution method, the process will continue into the contingency solution step unless the limiter that is being iterated on is a base case limitation. If the limiter is a base case limitation, a Transfer Limit will be reported the same as if using the Iterated Linear Step (IL) method.
 - o If using the [Iterated Linear \(IL\) then Full CTG](#) method and ramping post-contingency, e.g. the option to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions** is NOT checked, and the full transfer amount cannot be achieved, a transfer limitation will be reported highlighted in gray with the Iteratively Found field set to *RAMP_FAIL_IN_FULL_AFTER_CTG*. The Transfer Limit that is reported is the total amount that could be ramped before the power flow failed to converge.
 - o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. At the start of the iterated process, the base case generation and load values are stored. At each step where transfer ramping occurs, the generation and load values are returned to their base case levels and the transfer ramping is implemented from that point. As an example, take the situation where 1000 MW is ramped. The next Stepsize is calculated to be -100 MW. The total transfer amount achieved is then 900 MW. If ramping does not proceed in the same direction, generators that hit maximum limits when ramping out to 1000 MW will back off from these limits when ramping the -100 MW Stepsize. Ramping out to the total transfer amount of 900 MW in one step causes generators to hit their limits and stay there. If the ramping does not proceed in the same direction, ramping out by the total amount in one step and ramping by backing off the transfer will yield different results because some generators will no longer be at limits when ramping in steps. Ramping in the same direction is intended to prevent this discrepancy with generators at limits.
 - o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reverse limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active. When a reserve limit is found, the current iterations will stop and a transfer limit will be

reported with the reserve limit and the area, super area, or injection group that is at a limit. If a reserve limit is encountered during the step where all transfer limiters are iterated on together, the entire ATC process will stop. If a reserve limit is encountered during the step where a limiter is being iterated on individually, the iterations will only stop for that limiter; other limiters will be processed if they do not hit a reserve limit as well.

- At new operating point, go back to first step and repeat

As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the above process is repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process. Some limiters that will appear in the list of limiters are not valid for iterating on individually. These limiters include those indicating the full transfer amount could not be ramped in the initial process and those indicating that a reserve limitation has been met on the source or sink. These type of limiters will be skipped when choosing those to iterate on individually. Transfer limitations for transfer limits found by iterating on them individually will be highlighted in yellow in the results and the Iteratively Found field will be set to *YES*.

This method takes into account controller changes that occur as you ramp out to the transfer level, but still avoids the full simulation of contingencies.

This method can be applied between combinations of areas and super areas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

ATC Analysis Methods: Iterated Linear Step (IL) then Full CTG Solution

This method takes the solution process a step further than just the Iterated Linear (IL) method. Instead of using linear sensitivities to calculate the impact of contingencies on the transfer limits, full contingencies are implemented and solved with a full power flow solution. Be aware however, that this calculation method can be extremely slow.

The initial iteration process of (IL) then Full CTG Solution method starts by performing the steps outlined in the [Iterated Linear \(IL\)](#) process. As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the steps outlined in the [Iterated Linear \(IL\)](#) process are repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process. Some limiters that will appear in the list of limiters are not valid for iterating on individually. These limiters include those indicating the full transfer amount could not be ramped in the initial process and those indicating that a reserve limitation has been met on the source or sink. These type of limiters will be skipped when choosing those to iterate on individually.

Once the iterated process has been repeated for an individual limiter, there are two different ways that the processing of the full contingency can proceed. If choosing to **NOT Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, the contingency of each transfer limiter is then enforced in the power flow topology, and a power flow solution is performed to give a new contingency power flow solution to use as the base for each transfer limiter. If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency. The iterative process outlined in the [Iterated Linear \(IL\)](#) method is then repeated again for each individual transfer limiter with the specified contingency of each limiter actually in place in the power flow solution. If the contingency solution fails to converge, the transfer limitation will be reported highlighted in orange with the Iteratively Found field set to *CTG_FAIL_IN_FULL*, and the Transfer Limit that is reported will be the value determined from the iterated process performed on the individual limiter before the contingency was applied.

If choosing to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, after the iterated process is complete for an individual limiter, the contingency of each transfer limiter is fully implemented in the following steps:

- Store the system state following the iterated process on the individual limiter
 - At this point the accumulated transfer amount is the total amount that has been transferred following completion of the iterated process on the individual limiter.
 - The Stepsize is zero for the first iteration.
- Apply the contingency and perform full power flow solution
 - If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency.
 - If the option to **Iterate on failed contingency** is NOT checked and the power flow fails to solve, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in orange with the Iteratively Found field set to *CTG_FAIL_IN_FULL*.

The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution in which the contingency solved. If the contingency fails to solve immediately following the completion of the iterated process on the individual limiter without the contingency implemented, the Transfer Limit that is reported is the accumulated transfer amount achieved from that process without the contingency implemented.

- o If the option to **Iterate on failed contingency** IS checked and the power flow fails to solve, a new iteration loop is entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current Stepsize. To start the iterative process, the current Stepsize is reduced by half. The ramping is performed and the contingency is solved. The Stepsize will remain the same as long as the contingency can be solved. If the contingency fails to solve, the Stepsize will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original Stepsize, or the Stepsize becomes smaller than the Stepsize tolerance.
 - o An additional check is done if the contingency does solve during this process; the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the Stepsize will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved. During this entire process, the Stepsize is limited to be between zero and the Stepsize that was ramped when the contingency failed to solve for the first time. When the iterative process on the failed contingency completes, the iterations are done for this particular limiter. The Transfer Limit that is reported is the accumulated amount that is possible without overloading the monitored element and in which the contingency solves. The limiter will be reported highlighted in red with the Iteratively Found field set to *CTG_FAIL_ITERATED* if the monitored element is not loaded to its limit. If during this process, the monitored element is at its thermal limit and the contingency solves, the Iteratively Found field is set to *FULL* and the limiter will be highlighted in aqua.
 - o If the contingency fails to solve at any transfer level (base case failure), the limiter will be reported highlighted in purple with the Iteratively Found field set to *CTG_FAIL_IN_BASE*. The Transfer Limit that is reported is the accumulated transfer amount that was achieved from the iterated process on the individual limiter without the contingency implemented.
 - o If the power flow solution is successful, update the accumulated transfer with the Stepsize.
- Perform Single Linear Step on the individual limiter
 - o The Stepsize is the Transfer Limit found for the individual limiter. If the Stepsize is less than the specified **When iterating, Ignore Limiters below** value minus the accumulated transfer amount, it is set to zero.
- If $[\text{abs}(\text{Stepsize}) \leq \text{Tolerance}]$ or $[\text{Iterations} \geq 100]$ then stop
 - o If the maximum 100 number of iterations is reached, a transfer limitation will be reported highlighted in lime green with the Iteratively Found field set to *OSCILLATING* indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. The maximum number of iterations is a value that is set internally. It is unlikely that this limit will ever be reached because other mechanisms are in place to prevent oscillations.
 - o The iteration count is reset at the beginning of the contingency iteration process for each individual limiter.
- Restore system to the state stored in the first step

- o This step removes the contingency that was implemented so that the transfer ramping occurs prior to the contingency being applied.
- Implement transfer by amount of Stepsize and resolve power flow
 - o The power flow is solved according to the options set with the [Power Flow Solution Options](#).
 - o When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
 - o To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:
 - If Stepsize > 0 then LowBound = (Accumulated Transfer)
 - Else If Stepsize < 0 then HighBound = (Accumulated Transfer)
 - If Stepsize >= (HighBound - Accumulated Transfer)*0.7
 - Then Stepsize = (HighBound - Accumulated Transfer)*0.7
 - Else If Stepsize <= (LowBound - Accumulated Transfer)*0.7
 - Then Stepsize = (LowBound - Accumulated Transfer)*0.7
 - o During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes by reducing the stepsize by half. The stepsize reduction continues until a stepsize is found at which the power flow will solve. Once this stepsize is found, an attempt will be made to ramp to the original full stepsize using the smaller reduced stepsize. If at any point in this process a stepsize cannot be ramped, an iterative process will be used to reduce the stepsize so as much as possible of the full original stepsize can be ramped. An internal tolerance is used to prevent the stepsize from becoming too small.
 - o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. Additional details about this are provided in the steps of the [Iterated Linear \(IL\)](#) process.
 - o If the power flow fails to solve at this point without being able to implement the full Stepsize transfer, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in fuchsia with the Iteratively Found field set to *RAMP_FAIL_IN_FULL*. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution at which the contingency solved. The Transfer Limit reported does not indicate the portion of the Stepsize that could be ramped before the power flow fails to solve.
 - o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reserve limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active.

- When a reserve limit is found, the current iterations will stop and a transfer limit will be reported with the reserve limit and the area, super area, or injection group that is at a limit.
- Go back to second step and repeat

Transfer limitations found by implementing the full contingency solution are highlighted in aqua in the results with the Iteratively Found field set to *FULL*.

This method can be applied between combinations of areas and superareas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

When using this method, the PTDF and OTDF cutoff values are set to 0.01% internally by the ATC process when iterating on individual limiters. Once limiters are in the list as having acceptable distribution factors following the process where all limiters are iterated on as a whole, they should not be eliminated during the process when they are being iterated on individually. Actual implementation of the contingency as opposed to the linear determination of the impact of the contingency could lead to some large differences in the distribution factor post-contingency for some contingency scenarios. The only time that these differences could be a problem is if implementation of the contingency actually causes the monitored element to no longer be limiting, i.e. the distribution factor goes to zero. If this happens for any limiter, that limiter will be marked internally as having an infinite transfer amount, will show up in the results with a very large transfer limiter value, and will be excluded from the count of the number of limiters that have been iterated on. The next limiter in the list will then be processed until the number of **Transfer Limiters to Iterate on** has been met or there are no more limiters.

Simulator Automation Server (SimAuto) Overview

The SimAuto tool is only available if you have purchased the SimAuto add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the SimAuto version of Simulator.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPE](#)) add-ons. For more information see.

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on COM Objects and Automation Servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

Previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

Note: When Simulator is launched using the SimAuto object, any currently running instances of Simulator launched manually will be unaffected. SimAuto will launch a separate background instance of Simulator so that any manually opened instances of Simulator can continue to be used by the user.

Installing Simulator Automation Server

Note that previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.

If for some reason the registration fails, be sure you have the SimAuto add-on for Simulator and you have either run the application on the computer or run the command **pwrworld / regserver** at the command line. If instead you would like SimAuto to un-register itself as a COM object, run the command **pwrworld /unregserver** at the command line.

Including Simulator Automation Server Functions

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi, choose **Import Type Library...** from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click **Add**. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click **Open**.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click **Install** to include the PowerWorld Simulator Type Library.

Microsoft Visual Basic for Applications

- No additional tasks are necessary
- Importing Type Library still works (See [Including Functions for version 9](#)).

Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "... \powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

Matlab v.6.5 r.13

- No additional tasks are necessary.

Connecting to Simulator Automation Server

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Add **pwrworld_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto**
- Initialize the variable: **A := nil**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create**
- Perform function calls to the Simulator Automation Server: **Output := A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil**

Microsoft Visual Basic for Applications

Early Binding:

- To connect to the Simulator Automation Server, create the connection initializing the variable:

Dim A as New pwrworld.SimulatorAuto

Late Binding:

- Declare a variable globally or as part of another object or function: **Dim A As Object**
- To connect to the Simulator Automation Server, create the connection:

Set A = CreateObject("pwrworld.SimulatorAuto")

Both Early and Late Binding:

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction parameters**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing**
- If Type Library was imported, connection can also be achieved as in version 9 (See [Connecting to Simulator Automation Server in version 9](#)).

Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **IsimulatorAutoPtr *A**
- Declare a variable globally or as part of another object or function: **CLSID clsid**
- Declare a variable globally or as part of another object or function: **HRESULT hr**
- Obtain the class identifier (clsid) with the following command:

hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid)

- Initialize variable A: **A = new IsimulatorAutoPtr**
- To connect to the Simulator Automation Server, create the connection:

hr = A>CreateInstance(clsid, NULL, CLSCTX_SERVER)

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, release the reference:

hr = A>Release()

Matlab v.6.5 r.13

- To connect to the Simulator Automation Server, create the connection:

A = actxserver('pwrworld.SimulatorAuto')

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, delete the connection: **delete(A)**

Passing Data to the Simulator Automation Server

Passing Data to the Simulator Automation Server

All data to the Simulator Automation Server is passed by value rather than by reference (in pointer terminology, this corresponds to sending data instead of pointer to data; in Microsoft Visual Basic®, this corresponds to sending data ByVal instead of ByRef). This makes the marshalling of data between client software and the COM object much easier.

No Optional Parameters

There are no optional parameters for any of the Simulator Automation functions. All functions must be called with every argument filled.

Getting Data from the Simulator Automation Server

The Output Structure

Every function called on the SimulatorAuto object returns the same value, Output, which has a well-defined structure.

Output is of type **VARIANT**, and is an array of **VARIANTs**. Output is zero-indexed.

The first element *always* contains any errors occurring during execution. For those functions returning more than one element in the Output array (e.g. ListOfDevices), explanation is provided below when discussing the specific method.

Error Handling

As mentioned above, the first item in the Output VARIANT array, Output[0], contains any errors occurring during the function's execution. If no errors occurred during the function's execution, Output[0] will be set to an empty BSTR (string) represented in most languages by either "" or "".

Error Format

If an error string is returned, it will be in the following format:

[method name]: [error_explanation]

e.g. RunScriptCommand: Error occurred processing script command - check command syntax

Simulator Automation Server Functions

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[ChangeParametersSingleElement](#)

[ChangeParametersMultipleElement](#)

[ChangeParametersMultipleElementFlatInput](#)

[CloseCase](#)

[GetCaseHeader](#)

[GetParametersSingleElement](#)

[GetParametersMultipleElement](#)

[GetParametersMultipleElementFlatOutput](#)

[GetParameters](#) (for compatibility with version 9)

[ListOfDevices](#)

[ListOfDevicesAsVariantStrings](#)

[ListOfDevicesFlatOutput](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[GetFieldList](#)

[SaveState](#)

[LoadState](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

ChangeParameters Function

The ChangeParameters function has been replaced by the [ChangeParametersSingleElement](#) function. ChangeParameters can still be called as before, but will now just automatically call ChangeParametersSingleElement, and pass on the parameters to that function.

Function Prototype

ChangeParameters(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

ChangeParametersSingleElement Function

The ChangeParametersSingleElement function allows you to set a list of parameters for a single object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPE](#). This function is identical in setup to the [GetParametersSingleElement](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

Function Prototype

ChangeParametersSingleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables or [label](#) for the specific device, or the device cannot be identified.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

Output

ChangeParametersSingleElement only returns the first element in Output, the error string.

Notes

The ParameterList and Values arrays must be the same size, as each parameter must have a corresponding value to be assigned.

ChangeParametersSingleElement Sample Code

Borland® Delphi

```
Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the maximum number of iterations
// and the system base for the power flow simulations
ParamList := VarArrayCreate([1,2], varOleStr);
ParamList[1] := 'MaxItr';
ParamList[2] := 'SBase';

// ValueList is setup with 41 and 410 for MaxItr and SBase,
// respectively
ValueList := VarArrayCreate([1,2], varOleStr);
ValueList[1] := 41;
ValueList[2] := 410;

// Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
    ParamList, ValueList)
```

Microsoft® Visual Basic for Applications

```
' Set ParamList up to modify the maximum number of iterations
' and the system base for the power flow simulations
Dim ParamList As Variant
ParamList = Array("MaxItr", "Sbase")

' ValueList is setup with 41 and 410 for MaxItr and SBase,
' respectively
Dim ValueList As Variant
ValueList = Array(45, 90)

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement("Sim_Solution_Options", _
    ParamList, ValueList)
```

Matlab®

```
% Set ParamList up to modify the maximum number of iterations
% and the system base for the power flow simulations
```

```

ParamList = {'MaxItr' 'Sbase'};

% values is setup with 41 and 410 for MaxItr and SBase,
% respectively
values = [41 410];

% Convert the values matrix to a set of cells for passing
% through the COM interface
ValueList = num2cell(values);

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
ParamList, ValueList)

```

ChangeParametersMultipleElement Function

The ChangeParametersMultipleElement function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersSingleElement](#), but allows for modifying multiple elements with a single function call. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call ChangeParametersSingleElement multiple times.

Function Prototype

ChangeParametersMultipleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

ValueList : Variant A variant array storing arrays of variants. This is the difference between the multiple element and single element change parameter functions. This array stores a list of arrays of values matching the fields laid out in ParamList. You construct ValueList by creating an array of variants with the necessary parameters for each device, and then inserting each individual array of values into the ValueList array. SimAuto will pick out each array from ValueList, and calls ChangeParametersSingleElement internally for each array of values in ValueList.

Output

ChangeParametersMultipleElement only returns the first element in Output, the error string.

Notes

The ParameterList and each array of values stored in ValueList must be the same size, as each parameter must have a corresponding value to be assigned.

ChangeParametersMultipleElement Sample Code

Borland® Delphi

```
Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the MW output of generators
ParamList := VarArrayCreate([1,3], varOleStr);
ParamList[1] := 'BusNum';
ParamList[2] := 'GenID';
ParamList[3] := 'GenMW';

// ValueList is setup with MW values for generators at buses 1 and 2
ValueList := VarArrayCreate([1,2], varOleStr);
For k := 1 to 2 do begin
IndValueList := VarArrayCreate([1,3], varOleStr);
IndValueList[1] := k;
IndValueList[2] := '1';
IndValueList[3] := k*10;
ValueList[k] := IndValueList;
End;

// Make the ChangeParametersMultipleElement call
Output = SimAuto.ChangeParametersMultipleElement('Sim_Solution_Options', _
ParamList, ValueList)
```

Microsoft® Visual Basic for Applications

```
' Set ParamList up to modify the MW output of generators
Dim ParamList As Variant
ParamList = Array("BusNum", "GenID", "GenMW")

' ValueList is setup with MW values for generators at buses 1 and 2
Dim ValueList(2) As Variant
For k = 0 to 1
Dim IndValueList As Variant
IndValueList = Array(k+1,"1",(k+1)*10)
ValueList(k) = IndValueList
Next

' Make the ChangeParametersMultipleElement call
```

```
Output = SimAuto.ChangeParametersMultipleElement("Sim_Solution_Options", _  
ParamList, ValueList)
```

ChangeParametersMultipleElementFlatInput Function

The ChangeParametersMultipleElementFlatInput function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersMultipleElement](#), but uses a single dimensioned array of values as input instead of a multi-dimensioned array of arrays. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call [ChangeParametersSingleElement](#) multiple times. An additional advantage over [ChangeParametersMultipleElement](#) is that you can still take advantage of the speed improvement, even if the programming language you are using does not support multi-dimensioned arrays.

Function Prototype

ChangeParametersMultipleElement(ObjectType, ParamList, NoOfObjects, ValueList)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

ParamList : Variant A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

NoOfObjects You must pass an integer number of devices that are passing values for. SimAuto will automatically check that the number of parameters for each device (counted from ParamList) and the number of objects integer correspond to the number of values in value list (counted from ValueList.)

ValueList : Variant A variant array storing a list of variants. Value list can be an array with many values, as it is a single dimensioned array of all values for all devices that are being changed. The structure of the ValueList array is such that all of the parameters for the first object are listed first, then all parameters for the second object, and so on. The parameters must be in the same order as given in ParamList. In other words, your array would look like:

ValueList = Array(Obj1Param1, Obj1Param2, Obj2Param1, Obj2Param2, Obj3Param1, ..., ObjNParam1, ObjNParam2)

Output

ChangeParametersMultipleElementFlatInput only returns the first element in Output, the error string.

Notes

If the number of parameters given in ParamList multiplied by the number of objects passed does not equal the total number of values in ValueList, SimAuto will abort the function call.

ChangeParametersMultipleElementFlatInput Sample Code

Borland® Delphi

```
Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the MW output of generators
ParamList := VarArrayCreate([1,3], varOleStr);
ParamList[1] := 'BusNum';
ParamList[2] := 'GenID';
ParamList[3] := 'GenMW';

// ValueList is setup with MW values for generators at buses 1 and 2
NumObjects := 2;
NumFields := NumObjects * 3;
ValueList := VarArrayCreate([1,NumFields], varOleStr);
For k := 0 to 1 do begin
ValueList[3*k+1] := k;
ValueList[3*k+2] := '1';
ValueList[3*k+3] := k*10;
End;

// Make the ChangeParametersMultipleElementFlatInput call
Output = SimAuto.ChangeParametersMultipleElementFlatInput('Sim_Solution_Options', _
ParamList, NumObjects, ValueList)
```

Microsoft® Visual Basic for Applications

```
' Set ParamList up to modify the MW output of generators
Dim ParamList As Variant
ParamList = Array("BusNum", "GenID", "GenMW")

' ValueList is setup with MW values for generators at buses 1 and 2
NumObjects = 2
NumFields = NumObjects * 3
Dim ValueList(NumObjects) As Variant
For k = 0 to 1
ValueList(3*k+1) = k
ValueList[3*k+2] := '1';
ValueList[3*k+3] := k*10;
Next
```


' Make the ChangeParametersMultipleElementFlatInput call
Output = SimAuto.ChangeParametersMultipleElementFlatInput("Sim_Solution_Options", _
ParamList, NumObjects, ValueList)

CloseCase Function

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function.

Function Prototype

CloseCase()

Parameter Definitions

No parameters are passed.

Output

CloseCase returns only one element in Output—any errors which may have occurred when attempting to close the case.

CloseCase Function: Sample Code

Borland® Delphi

Output := SimAuto.CloseCase();

Microsoft® Visual Basic for Applications

Output = SimAuto.CloseCase()

Matlab®

Output = SimAuto.CloseCase

GetFieldList Function

The GetFieldList function is used to find all fields contained within a given object type.

Function Prototype

GetFieldList(ObjectType)

Parameter Definitions

ObjectType : String The type of object for which the fields are requested.

Output

GetFieldList returns two elements of the Output array. The first element, as with the other functions, returns any errors that might have occurred. The second element of the Output array contains an $n \times 4$ array of fields. The layout of this array is virtually identical to the output obtained by going to **Help > Export Case Object Fields** from the main menu of Simulator. The first column, corresponding to the $(n,0)$ column in the field array, specifies which fields are key fields for the object. The second column, $(n,1)$, contains the internal name of the field. The third column, $(n,2)$, contains the type of data stored in the string (e.g. String, Integer, Real). The fourth column, $(n,3)$, contains the display-friendly name of the field.

GetFieldList Function: Sample Code

Microsoft® Visual Basic for Applications

```
Dim objecttype As String
```

```
' Object type to obtain
```

```
objecttype = "branch"
```

```
' Make the GetField call
```

```
Output = SimAuto.GetFieldList(objecttype)
```

Matlab®

```
% Object type to obtain
```

```
objecttype = 'branch';
```

```
% Make the GetField call
```

```
Output = SimAuto.GetFieldList(objecttype);
```

GetParametersSingleElement Function

The GetParametersSingleElement function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParametersSingleElement function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

Function Prototype

GetParametersSingleElement(ObjectType, ParamList, Values)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

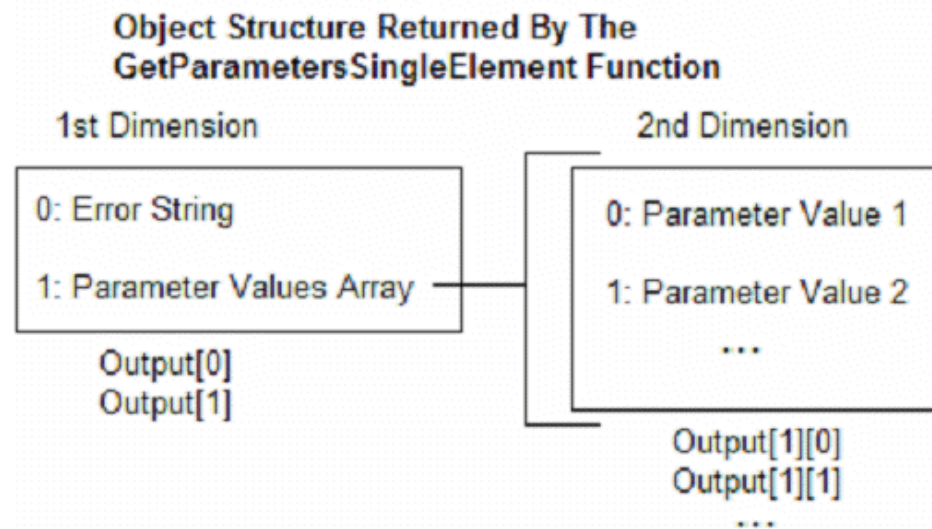
ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables or [label](#) for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array should be set to zero.

Output

GetParametersSingleElement returns both the first element in Output—containing any errors occurring during execution of the function—and a second element in Output. The second element returned in the Output structure is a one dimensional array containing the values corresponding to the fields specified in ParamList.

The Output structure of GetParametersSingleElement is shown in the following figure.



GetParametersSingleElement Function: Sample Code Borland® Delphi

```
// This example retrieves some parameters for bus 2 of the loaded
// case, using the GetParametersSingleElement function, as well as
// the old GetParameters function

procedure TMainForm.RunGPSEClick(Sender: TObject);
var
  Output : OLEVariant;
  FieldBusArray, ValueBusArray : OLEVariant;
  i : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';
  ValueBusArray := varArrayCreate([1,5],varOleStr);
  ValueBusArray[1] := 2; // To get parameters for bus 2
  ValueBusArray[2] := 0;
  ValueBusArray[3] := 0;
  ValueBusArray[4] := 0;
  ValueBusArray[5] := 0;

  // Gets parameters with GetParametersSingleElement function
  Output := SimAuto.GetParametersSingleElement('bus', FieldBusArray, ValueBusArray);
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersSingleElement ==');
      Memo1.Lines.Add('Value : Output[1][i]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));
      end;
      Memo1.Lines.Add("");
    end;
end;
```



```

end;

// Gets parameters with old function GetParameters
Output := SimAuto.GetParameters('bus', FieldBusArray, ValueBusArray);
if (string(Output[0]) <> "") then
  StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
else
  begin
    StatusBar1.Panels[1].Text := 'Parameters got.';
    Memo1.Lines.Add('== GetParameters ==');
    Memo1.Lines.Add('Value : Output[1][i]');
    for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
      Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));
    end;
    Memo1.Lines.Add("");
  end;
end;

```

GetParametersSingleElement Function: Sample Code Matlab®

```
% This example loads all buses in the case, and then gets
% some parameters of the last bus in the list

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus','');
    if ~(strcmp(output{1},''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for last bus of busarray
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    valuearray = [busarray(size(busarray,1)) '0' '0' '0'];
    valuelist = num2cell(valuearray);
    output = simauto.GetParametersSingleElement('bus',fieldarray,valuelist);
    if ~(strcmp(output{1},''))
        disp(output{1})
    else
        % Puts the buses in row vector busparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            busparam(i,1) = paramlist(i);
        end
        disp('Successful GetParameters for Bus')
```

```
disp(fieldarray)
disp(busparam)
end
end
```

GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel

```
Private Sub btnGetParametersSingleElement_Click()  
Dim objtype, filter As String  
Dim xlWB As Excel.Workbook  
Set xlApp = Excel.Application  
  
' Checks connection and open case  
' SimAuto and caseopen are global variables  
If Not SimAuto Is Nothing And caseopen Then  
    objtype = "bus"  
    Dim fieldArray As Variant  
    fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVVolt", _  
        "pwBusPUVolt", "pwBusAngle")  
    Dim ValueArray As Variant  
    ValueArray = Array(1, 0, 0, 0, 0)  
    output = SimAuto.GetParameters(objtype, fieldArray, ValueArray)  
    If output(0) <> "" Then  
        DisplayErrorMessage output(0)  
    Else  
        DisplayMessage "Successful GetParametersSingleElement"  
        ' Prepares additional worksheet  
        Set xlWB = xlApp.Workbooks.Add  
        ' Copies list of devices in worksheet  
        With xlWB  
            Sheets("sheet2").Activate  
            Sheets("sheet2").Name = "GetParametersSingleElement"  
            With Sheets("GetParametersSingleElement")  
                Dim i As Integer  
                Range(Cells(1, 5), Cells(200, 7)).Clear  
                Cells(1, 1) = "List of Devices for " + objtype + ":"  
                ' Setup fields as subheader  
                For i = LBound(fieldArray) To UBound(fieldArray)  
                    Cells(2, i + 1) = fieldArray(i)  
                Next i  
                ' Determine number of fields retrieved  
                Dim lowfld, highfld As Integer  
                lowfld = LBound(output(1), 1)  
                highfld = UBound(output(1), 1)
```

```
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
For i = lowfld To highfld
Cells(j + 3, i + 1) = output(1)(i)
Next i
End With
End With
End If
End If
End Sub
```

GetParametersMultipleElement Function

The GetParametersMultipleElement function is used to request the values of specified fields for a set of objects in the load flow case. The function can return values for all devices of a particular type, or can return values for only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case.

Function Prototype

GetParametersMultipleElement(ObjectType, ParamList, FilterName)

Parameter Definitions

ObjectType : String The type of object you are changing parameters for.

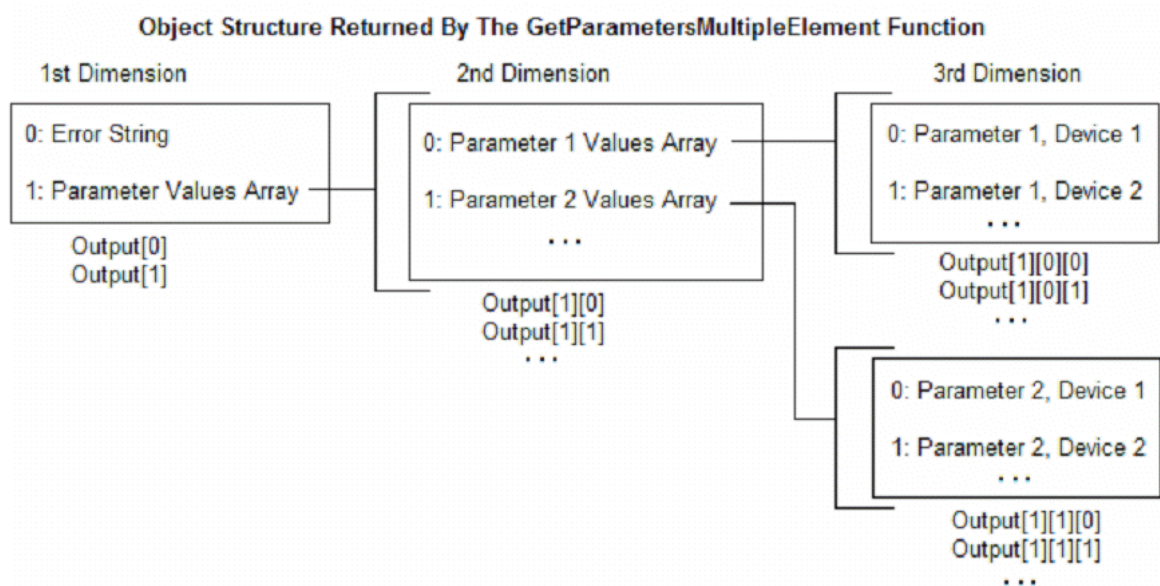
ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables or [label](#) for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, the server will default to returning all objects in the case of type ObjType.

Output

GetParametersMultipleElement returns a set of nested arrays containing the parameter values for the device type requested. The number of arrays of values returned depends on the number of fields in ParamList.

The Output structure of GetParametersMultipleElement is shown in the following figure.



As you can see, to access the first parameter value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number for the first bus would be stored at `Output[1][0][0]` after calling

`Output = GetParametersMultipleElement('Bus', fieldarray, '')`, and assuming that we have `fieldarray = Array(pwBusnum, pwBusName)`.

GetParametersMultipleElement Sample Code Borland® Delphi

```
// This example retrieves some parameters for all buses of the
// loaded case, using the GetParametersMultipleElement function

procedure TMainForm.RunGPMEClick(Sender: TObject);
var
  FieldBusArray : OLEVariant;
  i,j : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';

  // Gets parameters with Multiple Element function
  Output := SimAuto.GetParametersMultipleElement('bus', FieldBusArray, "");
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersMultipleElement ==');
      Memo1.Lines.Add('Value : Output[1][i][j]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        for j := VarArrayLowBound(Output[1][i],1) to VarArrayHighBound(Output[1][i],1)
        do begin
          Memo1.Lines.Add(FieldBusArray[i] + '(' + IntToStr(j) + ') : ' +
            string(Output[1][i][j]));
        end;
      end;
      Memo1.Lines.Add("");
    end;
end;
```

GetParametersMultipleElement Function: Sample Code Matlab®

```
% This example loads all buses in the case, and then gets
% some parameters of such buses

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus', '');
    if ~(strcmp(output{1}, ''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for all buses
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    output = simauto.GetParametersMultipleElement('bus', fieldarray, ' ');
    if ~(strcmp(output{1}, ''))
        disp(output{1})
    else
        % Puts the buses in matrix busesparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            for j=size(paramlist{i},2):size(paramlist{i},1)
                busesparam(j,i) = paramlist{i}(j);
            end
        end
        disp('Successful GetParametersMultipleElement')
```

```
disp(fieldarray)
disp(busesparam)
end
end
```

GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel

```
Private Sub btnGetParametersMultiple_Click()  
Dim objtype, filter As String  
Dim xlWB As Excel.Workbook  
Set xlApp = Excel.Application  
  
' Checks connection and open case  
' SimAuto and caseopen are global variables  
If Not SimAuto Is Nothing And caseopen Then  
    objtype = "bus"  
    filter = ""  
    Dim fieldArray As Variant  
    fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVVolt", _  
        "pwBusPUVolt", "pwBusAngle")  
    output = SimAuto.GetParametersMultipleElement(objtype, fieldArray, filter)  
    If output(0) <> "" Then  
        DisplayErrorMessage output(0)  
    Else  
        DisplayMessage "Succesful GetParametersSingleElement"  
        ' Prepares additional worksheet  
        Set xlWB = xlApp.Workbooks.Add  
        ' Copies list of devices in worksheet  
        With xlWB  
            Sheets("sheet2").Activate  
            Sheets("sheet2").Name = "GetParametersSingleElement"  
            With Sheets("GetParametersSingleElement")  
                Dim i, j As Integer  
                Range(Cells(1, 5), Cells(200, 7)).Clear  
                Cells(1, 1) = "List of Devices for " + objtype + ":"  
                ' Setup fields as subheader  
                For i = LBound(fieldArray) To UBound(fieldArray)  
                    Cells(2, i + 1) = fieldArray(i)  
                Next i  
                ' Determine number of fields retrieved  
                Dim lowfld, highfld As Integer  
                lowfld = LBound(output(1), 1)  
                highfld = UBound(output(1), 1)  
                ' Determine number of objects retrieved
```

```
Dim lowobj, highobj As Integer
lowobj = LBound(output(1)(lowkeyf), 1)
highobj = UBound(output(1)(lowkeyf), 1)
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
For i = lowfld To highfld
For j = lowobj To highobj
Cells(j + 3, i + 1) = output(1)(i)(j)
Next j
Next i
End With
End With
End If
End If
End Sub
```

GetParametersMultipleElementFlatOutput Function

This function operates the same as the [GetParametersMultipleElement](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the [GetParametersMultipleElement](#) topic. The function returns the parameter values for the device type requested, and the number of values returned depends on the ParamList and any [advanced filter](#) that impacts the number of devices returned.

The format of the output array is the following:

```
[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ...,  
Ob(n)Fld(m-1), Ob(n)Fld(m)]
```

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

GetParameters Function

This function is maintained in versions of Simulator later than version 9 for compatibility with Simulator version 9. This function is replaced by [GetParametersSingleElement](#). See also [GetParametersMultipleElement](#).

ListOfDevices Function

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParametersSingleElement](#) functions to process a group of devices.

Function Prototype

ListOfDevices(ObjType, filterName)

Parameter Definitions

ObjType : String The type of object for which you are acquiring the list of devices.

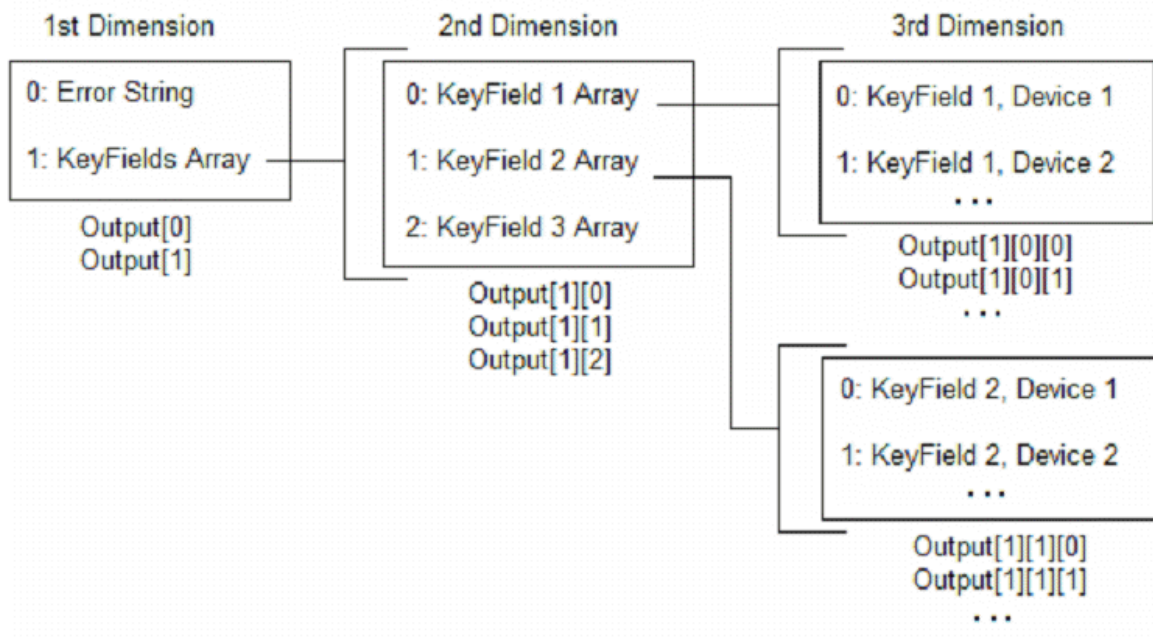
FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If the filter cannot be found, the server will default to returning all objects in the case of type ObjType.

Output

ListOfDevices returns a set of nested arrays containing the [key field](#) values for the device type requested. The number of arrays of values returned depends on the object type selected. For instance, buses have only one key field (the bus number) so calling ListOfDevices for buses will return only one array of values—the bus numbers. On the other hand, calling ListOfDevices for branches will return three arrays of values—the "From" bus, "To" bus, and ID—for each branch in the case meeting the specified filter.

The arrays containing the key field values for each device are arranged as shown in the following figure.

Object Structure Returned By The ListOfDevices Function



As you can see, to access the first key field value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number (which is the bus key field) for the first bus would be stored at `Output[1][0][0]` after calling `Output = ListOfDevices('Bus', '')`.

One unique limitation of the `ListOfDevices` function from other SimAuto functions is that this is the only function that returns the output as strongly typed variables. The bus numbers are always returned as Long Integers, and the Circuit ID values are returned as strings. This was actually an oversight during the design of SimAuto. In all other SimAuto functions, the values are returned as Variant types, with each value within the variant being a string. This was the intended operation for this function as well. Since the Automation Server interface was released with the errant inclusion of the `ListOfDevices` function, it could not be modified. Therefore, another function, `ListOfDevicesAsVariantStrings`, has been created. This function returns all values in variant variables, with each as a string within the variant type.

ListOfDevices Function: Sample Code for Borland® Delphi

Sample Code

```
// Runs Available Transfer Capability Routine
// Executes ATC Calculations among all areas
// and sends results to Excel
procedure TMainForm.RunATCClick(Sender: TObject);
var
  i, j, LowB, HighB : Integer;
  ValuesAreaArray : OLEVariant;
begin
  // Obtain all the areas
  Output := SimAuto.ListOfDevices('area', '');
  if (string(Output[0]) <> '') then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      ValuesAreaArray := Output[1][0];
      LowB := VarArrayLowBound(ValuesAreaArray, 1);
      HighB := VarArrayHighBound(ValuesAreaArray, 1);
      // Executes loop
      for i := LowB to HighB do
        for j := LowB to HighB do begin
          if (i <> j) then begin
            // Runs ATC calculations
            OutputATC := SimAuto.RunScriptCommand('entermode(atc); ' +
              'atcdetermine([Area ' + IntToStr(ValuesAreaArray[i]) +
              '], [Area ' + IntToStr(ValuesAreaArray[j]) + '])');
            if (string(OutputATC[0]) <> '') then
              StatusBar1.Panels[1].Text := 'Error: ' + string(OutputATC[0])
            else begin
              // Sends ATC results to Excel
              Output := SimAuto.SendToExcel('transferlimiter', '', 'all');
              if (string(Output[0]) <> '') then
                StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
              else
                StatusBar1.Panels[1].Text := 'ATC calculations done.';
            end;
          end;
        end;
    end;
end;
```

end;
end;
end;

ListOfDevices Function: Sample Code for Matlab®

Sample Code

```
%A list of branches is desired, without using any filter
DeviceType = 'Branch';
FilterName = '';
%Execute the ListOfDevices command
Output = SimAuto.ListOfDevices(DeviceType,FilterName);
%If the first cell in SimAutoOutput ~= "", then that means an error
%occurred.
if ~(strcmp(SimAutoOutput{1},''))
    disp(SimAutoOutput{1})
else
    %Otherwise, no errors. Display the branch information.
    disp('ListOfDevices successful')

    %Devicelist1 contains the From: bus number
    %Devicelist2 contains the To: bus number
    %Devicelist3 contains the Bus identifier
    devicelist1 = double(transpose(SimAutoOutput{2}{1}));
    devicelist2 = double(transpose(SimAutoOutput{2}{2}));
    devicelist3 = SimAutoOutput{2}{3};

    %If the device list is greater than 25, don't bother attempting to
    %display it on the screen.
    if (size(devicelist1,1) > 25)
        disp('Device list exceeds 25; use "devicelist" to manage list of devices')
    else
        %Otherwise, display the branches' information.
        disp(DeviceType)
        disp('From/To/Identifier')
        for counter = 1:size(devicelist1,1)
            %num2str converts the numbers within devicelist1 and
            %devicelist2 to strings for output with disp(). the char()
            %function is called on devicelist3's members because the
            %SimAuto object returns a character array (as opposed to a
            %properly Matlab-format string) and this array must be
            %converted to a Matlab-format string.
            disp([num2str(devicelist1(counter)) ' ' ...
```

```
num2str(devicelist2(counter)) ' ' ...  
char(devicelist3(counter))])  
end  
end  
end
```

ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel

Sample Code

```
Private Sub DisplayMessage(ByVal SentText As String)
    TextBox.Text = TextBox.Text + SentText + vbCrLf + vbCrLf
End Sub
```

```
Private Sub btnListOfDevices_Click()
    Dim objtype, filter As String
    Dim xlWB As Excel.Workbook
    Set xlApp = Excel.Application

    ' Checks connection and open case
    ' SimAuto and caseopen are global variables
    If Not SimAuto Is Nothing And caseopen Then
        objtype = "branch"
        filter = ""
        output = SimAuto.ListOfDevices(objtype, filter)
        If output(0) <> "" Then
            DisplayMessage output(0)
        Else
            DisplayMessage "Successful List Of Devices"
        ' Prepares additional worksheet
        Set xlWB = xlApp.Workbooks.Add
        ' Copies list of devices in worksheet
        With xlWB
            Sheets("sheet1").Activate
            Sheets("sheet1").Name = "ListOfDevices"
            With Sheets("ListOfDevices")
                Dim i, j As Integer
                Range(Cells(1, 5), Cells(200, 7)).Clear
                Cells(1, 1) = "List of Devices for " + objtype + ":"
                Cells(2, 1) = "From Bus Num"
                Cells(2, 2) = "To Bus Num"
                Cells(2, 3) = "ID"
            ' Determine number of key fields retrieved
            Dim lowkeyf, highkeyf As Integer
            lowkeyf = LBound(output(1), 1)
            highkeyf = UBound(output(1), 1)
        End With
    End If
End Sub
```

```

DisplayMessage "Number of Key Fields: " + Str(lowkeyf) + Str(highkeyf)
' Determine number of objects retrieved
Dim lowobj, highobj As Integer
lowobj = LBound(output(1)(lowkeyf), 1)
highobj = UBound(output(1)(lowkeyf), 1)
DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
For i = lowkeyf To highkeyf
For j = lowobj To highobj
Cells(j + 3, i + 1) = output(1)(i)(j)
Next j
Next i
End With
End With
End If
End If
End Sub

```

ListOfDevicesAsVariantStrings Function

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned as Variants of type String. The ListOfDevices function was errantly released returning the values strongly typed as Integers and Strings directly, whereas all other SimAuto functions returned data as Variants of type String. This function was added to also return the data in the same manner. This solved some compatibility issues with some software languages.

ListOfDevicesFlatOutput Function

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the ListOfDevices topic. The function returns the key field values for the device, typically in the order of bus number 1, bus number 2 (where applicable), and circuit identifier (where applicable). These are the most common key fields, but some object types do have other key fields as well.

The format of the output array is the following:

```
[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ...,  
Ob(n)Fld(m-1), Ob(n)Fld(m)]
```

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

LoadState Function

LoadState is used to load the system state previously saved with the [SaveState](#) function. Note that LoadState will not properly function if the system topology has changed due to the addition or removal of the system elements.

Function Prototype

LoadState()

Parameter Definitions

No parameters are passed.

Output

LoadState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

LoadState Function: Sample Code

Microsoft® Visual Basic for Applications

```
' Make the LoadState call  
Output = SimAuto.LoadState()
```

Matlab®

```
% Make the LoadState call  
Output = SimAuto.LoadState();
```

OpenCase Function

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). This is equivalent to opening a file using the **File > Open Case** menu option in Simulator.

Function Prototype

OpenCase(FileName)

Parameter Definitions

FileName : String The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

Output

OpenCase returns only one element in Output—if the file cannot be found or an error occurs while reading the file.

OpenCase Function: Sample Code

Borland® Delphi

```
Output := SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb');
if (string(Output[0]) <> '') then
  StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0]);
else
begin
  StatusBar1.Panels[1].Text := 'Open Case successful.';
  // Perform activities with opened case
end;
```

Microsoft® Visual Basic for Applications

```
Output = SimAuto.OpenCase("c:\simauto\examples\b7opf.pwb")
If output(0) <> "" Then
  MsgBox(output(0))
Else
  ' Perform activities with the opened case
End If
```

Matlab®

```
Output = SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb')
%If the first cell in Output ~= "", then that means an error
%occurred.
if ~(strcmp(Output{1}, ''))
  disp(Output{1})
else
  %Otherwise, no errors. Perform activities.
  disp('Open Case successful')
end
```

ProcessAuxFile Function

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator

Function Prototype

ProcessAuxFile(FileName)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

Output

ProcessAuxFile returns only one element in Output—any errors which may have occurred when attempting to load the file.

ProcessAuxFile Function: Sample Code

Microsoft® Visual Basic for Applications

Dim filename As String

' Setup name of aux file to run
filename = "c:\b7opf_ctglist.aux"

' Make the processAuxFile call
Output = SimAuto.ProcessAuxFile(filename)

Matlab®

% Setup name of aux file to run
filename = 'c:\b7opf_ctglist.aux';

% Make the processAuxFile call
Output = SimAuto.ProcessAuxFile(filename);

RunScriptCommand Function

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

Function Prototype

RunScriptCommand(Statements)

Parameter Definitions

Statements : String The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

Output

RunScriptCommand returns only one element in Output—any errors which may have occurred when attempting to load or run the auxiliary file.

RunScriptCommand Function: Sample Code

Microsoft® Visual Basic for Applications

Dim scriptcommand As String

' Set script command to cause Simulator to enter Run Mode
scriptcommand = "EnterMode(RUN)"

' Make the RunScriptCommand call
Output = SimAuto.RunScriptCommand(scriptcommand);

' Set script command to cause Simulator to perform a single,
' standard solution
scriptcommand = "SolvePowerFlow(RECTNEWT)"

' Make the RunScriptCommand call
Output = SimAuto.RunScriptCommand(scriptcommand);

Matlab®

% Set script command to cause Simulator to enter Run Mode
scriptcommand = 'EnterMode(RUN)';

% Make the RunScriptCommand call
Output = SimAuto.RunScriptCommand(scriptcommand);

% Set script command to cause Simulator to perform a single,
% standard solution
scriptcommand = 'SolvePowerFlow(RECTNEWT)';

% Make the RunScriptCommand call
Output = SimAuto.RunScriptCommand(scriptcommand);

SaveCase Function

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file.

Function Prototype

SaveCase(FileName, EString, FileType, Overwrite)

Parameter Definitions

FileName : String The name of the file you wish to save as, including file path.

FileType : String A string indicating the format of the written case file. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)
"PTI24" PTI version 24 (raw)
"PTI25" PTI version 25 (raw)
"PTI26" PTI version 26 (raw)
"PTI27" PTI version 27/28 (raw)
"PTI29" PTI version 29 (raw)
"PTI30" PTI version 30 (raw)
"PTI31" PTI version 31 (raw)
"GE" GE PSLF (epc)
"IEEE" IEEE common format (cf)
"PWB70" PowerWorld Binary version 7.0 (pwb)
"PWB" PowerWorld Binary (most recent) (pwb)

Overwrite : Boolean A Boolean value which indicates whether to overwrite a file if FileName already exists. If Overwrite is set to false and the file specified by FileName already exists, SaveCase will return an error message and do nothing to the file.

Output

SaveCase returns only one element in Output—any errors which may have occurred when attempting to save the case.

SaveCase Function: Sample Code

Microsoft® Visual Basic for Applications

' Save the case as a PWB file

```
Output = SimAuto.SaveCase("c:\b7opfcopy.pwb", "PWB", true)
```

' Save the case as a PTI file

```
Output = SimAuto.SaveCase("c:\b7opfcopy.raw", "PTI", true)
```

Matlab®

% Setup name of PWB file to write

```
filenamepwb = 'c:\b7opfcopy.pwb';
```

% Setup name of PTI file to write

```
filenamepti = 'c:\b7opfcopy.raw';
```

% Make the SaveCase call for the PWB file

```
Output = SimAuto.SaveCase(filenamepwb, 'PWB', true);
```

% Make the SaveCase call for the PTI file

```
Output = SimAuto.SaveCase(filenamepti, 'PWB', true);
```

SaveState Function

SaveState is used to save the current state of the power system. This can be useful if you are interested in comparing various cases, much as the [Difference Flows](#) feature works in the Simulator application.

Function Prototype

SaveState()

Parameter Definitions

No parameters are passed.

Output

SaveState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

SaveState Function: Sample Code

Microsoft® Visual Basic for Applications

```
' Make the SaveState call  
Output = SimAuto.SaveState()
```

Matlab®

```
% Make the SaveState call  
Output = SimAuto.SaveState();
```

SendToExcel Function

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed.

Function Prototype

SendToExcel(ObjectType, FilterName, FieldList)

Parameter Definitions

ObjectType : String A string describing the type of object for which your are requesting data.

FilterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

FieldList : Variant This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Output

SendToExcel returns only one element in Output—any errors which may have occurred when attempting to execute the function.

SendToExcel Function: Sample Code

Microsoft® Visual Basic for Applications

Dim FieldList As Variant

' Setup fieldlist to send the bus number, gen id and gen agc to Excel

FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")

' Make the SendToExcel call

' By specifying the parameter FieldList, only the three fields

' for each generator will be returned

Output = SimAuto.SendToExcel("gen", "", "FieldList")

' Sending the string "all" instead of a fieldlist array

' writes all predefined fields to the Excel spreadsheet

Output = SimAuto.SendToExcel("gen", "", "all")

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Matlab®

% Setup fieldlist to send the bus number, gen id and gen agc to Excel

fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };

% Make the SendToExcel call

Output = SimAuto.SendToExcel('gen', "", fieldlist);

% Sending the string 'all' instead of a fieldlist array

% writes all predefined fields to the Excel spreadsheet

Output = SimAuto.SendToExcel('gen', "", 'all');

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

TSGetContingencyResults Function

TSGetCongencyResults function

The TSGetContingencyResults function is used to read transient stability results into an external program (i.e. Matlab or VB) using SimAuto.

This function is analogous to the script command TSGetResults, where rather than saving out results to a file, the results are passed back directly to the SimAuto COM object and may be further processed by an external program. As with TSGetResults, this function should only be used after the simulation is run (for example, use this after running script commands TSSolveAll or TSSolve).

Function Prototype

TSGetContingencyResults(CtgName, ObjFieldList, StartTime, StopTime)

Parameter Definitions

CtgName : String The contingency to obtain results from. Only one contingency be obtained at a time.

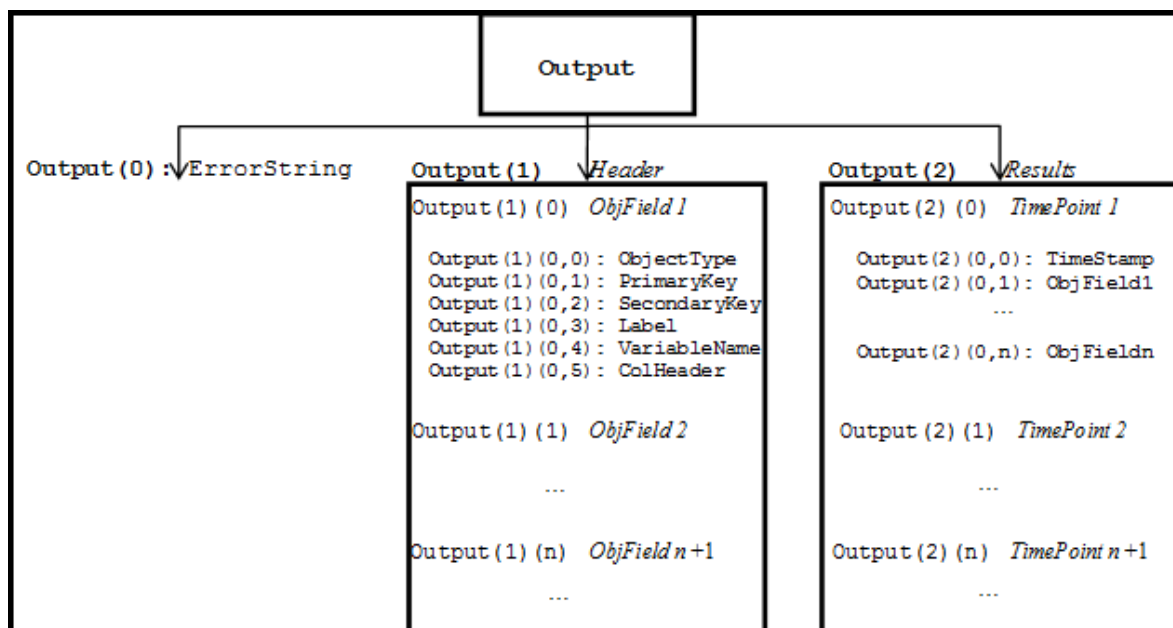
ObjFieldList: Variant A variant array of strings which may contain plots, subplots, or individual object/field pairs specifying the result variables to obtain.

StartTime : String The time in seconds in the simulation to begin retrieving results. If not specified, the start time of the simulation is used.

StopTime : String The time in seconds in the simulation to stop retrieving results. If not specified, the end time of the simulation is used.

Output

The SimAuto output for this function is a Variant which contains three levels. Output(0) displays an error message, if any. Output(1) displays the header which describes the variables that were saved out. Output(2) displays the time series data (numerical results from simulation), corresponding to the variables described in the header. MATLAB indexing begins at 1 instead of 0, so add one to all the indices shown when using MATLAB.



TSGetContingencyResults Sample Code

Sample code (VB):

```
Set mySimAuto = New pwrworld.SimulatorAuto
Output = mySimAuto.OpenCase("G:\wsc_9busCacheTest.pwb")
If Output(0) <> "" Then
    DisplayErrorMessage Output(0)
End If

Dim objFieldList As Variant
objFieldList = Array("Plot 'Gen_Rotor Angle'", "Bus 4 | frequency")
Output = mySimAuto.TSGetContingencyResults("ctgname1", objFieldList, "0.0", "10.0")
If Output(0) <> "" Then
    DisplayErrorMessage Output(0)
End If
Set SimAuto = Nothing
```

Locals		
VBAProject.Sheet1.CommandButton1_Click		
Expression	Value	Type
Me		Sheet1/Sheet1
Output		Variant/Variant(0 to 1)
Output(0)	""	Variant/String
Output(1)		Variant/String(0 to 3)
Output(1)(0)		String(0 to 5)
Output(1)(0,0)	"Generator "	String
Output(1)(0,1)	" 2 1 "	String
Output(1)(0,2)	" 'Bus 2_ 18.00' 1 "	String
Output(1)(0,3)	""	String
Output(1)(0,4)	"TSGenDelta "	String
Output(1)(0,5)	"Rotor Angle"	String
Output(1)(1)		String(0 to 5)
Output(1)(1,0)	"Generator "	String
Output(1)(1,1)	" 3 1 "	String
Output(1)(1,2)	" 'Bus 3_ 13.80' 1 "	String
Output(1)(1,3)	""	String
Output(1)(1,4)	"TSGenDelta "	String
Output(1)(1,5)	"Rotor Angle"	String
Output(1)(2)		String(0 to 5)
Output(1)(2,0)	"Generator "	String
Output(1)(2,1)	" 1 1 "	String
Output(1)(2,2)	" 'Bus1_ 16.50' 1 "	String
Output(1)(2,3)	""	String
Output(1)(2,4)	"TSGenDelta "	String
Output(1)(2,5)	"Rotor Angle"	String
Output(1)(3)		String(0 to 5)
Output(2)		Variant/String(0 to 1)
Output(2)(0)		String(0 to 4)
Output(2)(0,0)	"0"	String
Output(2)(0,1)	"58.2813873291016"	String
Output(2)(0,2)	"50.7263145446777"	String
Output(2)(0,3)	"3.23561811447144"	String
Output(2)(0,4)	"60"	String
Output(2)(1)		String(0 to 4)
Output(2)(1,0)	"0.008333"	String
Output(2)(1,1)	"58.2813873291016"	String
Output(2)(1,2)	"50.7263145446777"	String
Output(2)(1,3)	"3.23561811447144"	String
Output(2)(1,4)	"60"	String
Output(2)(2)		String(0 to 4)
Output(2)(2,0)	"0.016667"	String
Output(2)(2,1)	"58.2813873291016"	String
Output(2)(2,2)	"50.7263145446777"	String
Output(2)(2,3)	"3.23561811447144"	String
Output(2)(2,4)	"60"	String
Output(2)(3)		String(0 to 4)
Output(2)(4)		String(0 to 4)
Output(2)(5)		String(0 to 4)

Sample code (Matlab):

```
%Initialize SimAuto object
SimAuto = actxserver('pwrworld.SimulatorAuto');
%Initialize SimAutoOutput which is used to store the output of every method call
clear SimAutoOutput;
SimAutoOutput = {};
%Open the case
SimAutoOutput = SimAuto.OpenCase('G:\wsc_9busCacheTest.pwb');
if ~(strcmp(SimAutoOutput{1},''))
disp(SimAutoOutput{1})
else
disp('OpenCase successful')
end
%%
% Here we get the results for all of the angles directly into Matlab via SimAuto
%%
newCtgName = 'ctgName';
objFieldList = {'Plot "Gen_Rotor Angle"' };
SimAutoOutput = SimAuto.TSGetContingencyResults(newCtgName, objFieldList , '0.0', '10.0');
if ~(strcmp(SimAutoOutput{1},''))
disp(SimAutoOutput{1})
else
disp('GetTSResultsInSimAuto successful')

%Get the results
localResults = SimAutoOutput{3};

%Get the header variables to use for plot labels
Header = SimAutoOutput{2};

% Convert a matrix of strings into a matrix of numbers and plot them
localResultsMat = str2double(localResults);
plot(localResultsMat(:,1), localResultsMat(:,2:size(localResultsMat,2))));
title(['Transient stability results for ', newCtgName]);
TimeSeriesNames = Header([4:size(Header,1)],2);
legend(TimeSeriesNames); % these labels come from the HEADER
xlabel('time')
ylabel('Generator rotor angle (degrees)');
end
```

%Delete (close) the COM object.
delete(SimAuto);

Array Editor - SimAutoOutput{2,1}

Stack: tsScript_GSOGenBusFaults_Clean

	1	2	3	4	5	6
1	'Generator'	'53 1 '	'BLT138_ ...'		TSGenDe...	'Rotor Angle'
2	'Generator'	'54 1 '	'BLT69_ ...'		TSGenDe...	'Rotor Angle'
3	'Generator'	'48 1 '	'BOB69_ ...'		TSGenDe...	'Rotor Angle'
4	'Generator'	'28 1 '	'JO345_3...'		TSGenDe...	'Rotor Angle'
5	'Generator'	'28 2 '	'JO345_3...'		TSGenDe...	'Rotor Angle'
6	'Generator'	'44 1 '	'LAUF69...'		TSGenDe...	'Rotor Angle'
7	'Generator'	'50 1 '	'ROGER...'		TSGenDe...	'Rotor Angle'
8	'Generator'	'31 1 '	'SLACK3...'		TSGenDe...	'Rotor Angle'
9	'Generator'	'14 1 '	'WEBER...'		TSGenDe...	'Rotor Angle'

SimAutoOutput × SimAutoOutput{2,1} × SimAutoOutput{3,1} ×

Array Editor - SimAutoOutput{3,1}

Stack: tsScript_GSOGenBusFaults_Clean

	1	2	3	4	5	6	7	8	9	10
1	'0'	'33.93003...'	'20.00261...'	'6.293289...'	'48.65386...'	'48.65386...'	'51.31787...'	'15.61439...'	'19.06124...'	'13.32028...'
2	'0.008333'	'33.93003...'	'20.00261...'	'6.293289...'	'48.65386...'	'48.65386...'	'51.31787...'	'15.61439...'	'19.06124...'	'13.32028...'
3	'0.016666'	'33.93003...'	'20.00261...'	'6.293290...'	'48.65387...'	'48.65387...'	'51.31786...'	'15.61439...'	'19.06124...'	'13.32028...'
4	'0.024999'	'33.93003...'	'20.00260...'	'6.293292...'	'48.65387...'	'48.65387...'	'51.31785...'	'15.61440...'	'19.06124...'	'13.32028...'
5	'0.033332'	'33.93004...'	'20.00260...'	'6.293293...'	'48.65387...'	'48.65387...'	'51.31784...'	'15.61440...'	'19.06124...'	'13.32028...'
6	'0.041665'	'33.93004...'	'20.00259...'	'6.293296...'	'48.65388...'	'48.65388...'	'51.31783...'	'15.61440...'	'19.06124...'	'13.32028...'
7	'0.049998'	'33.93004...'	'20.00259...'	'6.293299...'	'48.65388...'	'48.65388...'	'51.31782...'	'15.61440...'	'19.06124...'	'13.32028...'
8	'0.058331'	'33.93004...'	'20.00258...'	'6.293302...'	'48.65388...'	'48.65388...'	'51.31781...'	'15.61441...'	'19.06125...'	'13.32028...'
9	'0.066664'	'33.93005...'	'20.00257...'	'6.293305...'	'48.65389...'	'48.65389...'	'51.31779...'	'15.61441...'	'19.06125...'	'13.32028...'
10	'0.074997'	'33.93006...'	'20.00256...'	'6.293308...'	'48.65390...'	'48.65390...'	'51.31777...'	'15.61441...'	'19.06125...'	'13.32028...'

SimAutoOutput × SimAutoOutput{2,1} × SimAutoOutput{3,1} ×

WriteAuxFile Function

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file.

Function Prototype

WriteAuxFile(FileName, FilterName, ObjectType, EString, ToAppend, FieldList)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file you wish to save.

FilterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

ObjectType : String A string describing the type of object for which you are requesting data.

ToAppend : Boolean If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If ToAppend is *False* and the file already exists, WriteAuxFile will return an error message and do nothing to the file.

FieldList : Variant This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Output

WriteAuxFile returns only one element in Output—any errors which may have occurred when attempting to execute the function.

WriteAuxFile Function: Sample Code

Microsoft® Visual Basic for Applications

Dim FieldList As Variant

Dim auxfilename As String

' Setup FieldList to send the bus number, gen id and gen agc

FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")

' Aux file to write to

auxfilename = "c:\businfo.aux"

' Make the WriteAuxFile call

' By specifying the parameter FieldList, only the three fields

' for each generator will be returned

Output = SimAuto.WriteAuxFile(auxfilename, "", "gen", true, FieldList)

' Sending the string "all" instead of the FieldList array

' writes all predefined fields to the Excel spreadsheet

Output = SimAuto.SendToExcel(auxfilename, "", "gen", true, "all")

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Matlab®

% Setup FieldList to send the bus number, gen id and gen agc

fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };

% Aux file to write to

auxfilename = 'c:\businfo.aux';

% Make the WriteAuxFile call

Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, FieldList);

% Sending the string 'all' instead of the FieldList array

% writes all predefined fields to the .aux file

Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, 'all');

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

Simulator Automation Server Properties

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

[CurrentDir](#)

[ProcessID](#)

[RequestBuildDate](#)

ExcelApp Property

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within its own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object. The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

Examples

Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

Microsoft Visual Basic

- **Dim MyExc As Variant**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

ExcelApp Property: Sample Code

Microsoft® Visual Basic for Applications

```
Dim ExcelObject As Variant
If Not (IsEmpty(SimAuto.ExcelApp)) Then
    Set ExcelObject = SimAuto.ExcelApp
    ExcelObject.DisplayAlerts = False
    ExcelObject.Quit
Else
    MsgBox("Attempted to obtain Excel COM object from " + _
    "Simulator, but there is not one open.")
End If
```

Matlab®

MATLAB is currently unable to handle COM object properties returning COM objects themselves, such as the ExcelApp property of Simulator.

CurrentDir Property

The CurrentDir property of the [Simulator Automation Server](#) allows you to retrieve or set the working directory for the currently running SimulatorAuto process. This is most useful if using relative filenames (e.g. "relativename.aux" versus "C:\Program Files\PowerWorld\Working\abosultename.aux") when specifying files.

CurrentDir Property: Sample Code

Microsoft® Visual Basic for Applications

' Display the current directory

```
MsgBox(SimAuto.CurrentDir)
```

' Set the current directory to c:\

```
SimAuto.CurrentDir = "c:\"
```

Matlab®

% Display the current directory

```
disp(SimAuto.CurrentDir)
```

% Set the current directory to c:\

```
SimAuto.CurrentDir = 'c:\';
```

ProcessID Property

The ProcessID property of the [Simulator Automation Server](#) allows you to retrieve the process ID of the currently running SimulatorAuto process, as can also be seen through the Task Manager in Windows® NT 4, 2000 and XP. This information can be useful if a forced shutdown of the SimulatorAuto object is needed, as all calls to the SimulatorAuto object are synchronous. This means the SimulatorAuto object will not be destroyed until all calls, no matter the time of execution, have completed.

ProcessID Property: Sample Code

Microsoft® Visual Basic for Applications

```
' Display the process ID  
MsgBox(SimAuto.ProcessID)
```

Matlab®

```
% Display the process ID  
disp(num2str(SimAuto.ProcessID))
```

RequestBuildDate Property

The RequestBuildDate property of the [Simulator Automation Server](#) allows you to retrieve the build date of the PowerWorld Simulator executable currently running with the SimulatorAuto process. The property returns an integer value that represents a date. This information is useful for verifying the release version of the executable.

PowerWorld Object Variables

The ability to access power system data for different objects through various [Simulator Automation Server Functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParametersSingleElement](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

Examples of Field Variables

BusGenMW

BusNum

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Case Object Fields...** The list of fields can either be saved to a text file or sent to Excel. The list will consist of the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

Examples of Field Variables in Listing

BusGenMW Real 'Bus Gen MW'

* BusNum Integer 'Bus Number' (* denotes key field in text file)

ATC_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is BusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as BusPUVolt:0, and the "To" bus voltage would be represented as BusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.

Simulator Automation Server (version 9)

NOTE: The function calls for version 9 of SimAuto have become obsolete in version 10. The reason for the change was due to problems that arose with the structure of the functions in certain programming languages. The function structures have been modified for version 10 of Simulator. If you have code written that uses version 9 SimAuto function structures, it is fairly straightforward to convert the function calls from version 9 to version 10. Please see the updated section on the [Simulator Automation Server](#) for version 10. If you still have questions, please contact PowerWorld Corporation.

The PowerWorld® Automation Server is only available to customers who have purchased the SimAuto add-on for PowerWorld® Simulator. The PowerWorld Simulator Automation Server is intended for enabling a PowerWorld customer with the ability to access PowerWorld Simulator functionality from within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various different programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, and Microsoft® Visual Basic, just to name a few. For more information on COM and Automation servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

Installing Simulator Automation Server (version 9)

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.

Including Simulator Automation Server Functions (version 9)

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi 5, choose Import Type Library... from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click Add. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click Install to include the PowerWorld Simulator Type Library.

Microsoft Visual Basic

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In VB, choose **References...** from the **Tools** menu. Find and choose pwrworld Library from the list of references.
- If pwrworld Library is not in the list, click Browse. Change the file type to *.exe, find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- Click OK to install the Simulator Type Library reference.

Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "... \powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

Connecting to Simulator Automation Server (version 9)

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

Borland Delphi 5

- Add **pwrworld_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto.**
- Initialize the variable: **A := nil.**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil.**

Microsoft Visual Basic

- Declare a variable globally or as part of another object or function: **Public A As SimulatorAuto.**
- To connect to the Simulator Automation Server, create the connection: **Set A = New SimulatorAuto.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction parameters.**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing.**

Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **IsimulatorAutoPtr *A.**
- Declare a variable globally or as part of another object or function: **CLSID clsid.**
- Declare a variable globally or as part of another object or function: **HRESULT hr.**
- Obtain the class identifier (clsid) with the following command: **hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid).**
- Initialize variable A: **A = new IsimulatorAutoPtr.**
- To connect to the Simulator Automation Server, create the connection: **hr = A>CreateInstance(clsid, NULL, CLSCTX_SERVER).**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**

- To close the connection to the Simulator Automation Server, release the reference: **hr = A>Release()**.

Simulator Automation Server Properties (version 9)

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

ExcelApp Property (version 9)

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within their own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object.

The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

Examples

Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

Microsoft Visual Basic

- **Dim MyExc As Object**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

Simulator Automation Server Functions (version 9)

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[CloseCase](#)

[GetParameters](#)

[ListOfDevices](#)

[LoadContingencies](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

ChangeParameters Function (version 9)

The ChangeParameters function allows you to set a list of parameters for a specific object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPF](#). This function is identical in setup to the [GetParameters](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

ChangeParameters(tObjectType, ParamList, Values, EString)

Parameter Definitions

tObjectType : String The type of object you are changing parameters for. No default.

ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. No Default.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example

ChangeParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], [1, "1", "Yes"], EString)

This function call will change the AGC Status of bus number one, generator ID number 1, to "Yes", meaning the generator will be included in AGC if it's area is on AGC control.

CloseCase Function (version 9)

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function. An error will be returned through the EString parameter if an error occurred while trying to close the case.

CloseCase(EString)

Parameter Definitions

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

GetParameters Function (version 9)

The GetParameters function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParameters function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

GetParameters(tObjectType, ParamList, Values, EString)

Parameter Definitions

TobjectType : String The type of object you are changing parameters for. No default.

ParamList : Variant A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator. No Default.

Values : Variant A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array can initially be filled with either empty strings or zeroes, as the values will be replaced when the function is processed. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example

GetParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], ValArray = [1, "1", ""], EString)

This function call will return the AGC Status of bus number one, generator ID number 1, in the ValArray variable. Note that the generator bus number and ID had to be passed into the function, while the remaining field could be assigned an empty string. If the bus number and ID had not been passed, an error would have been returned since Simulator would not be able to identify a generator. With a valid bus number and ID for the generator, the ValArray variable would have returned an array of the following format: ValArray = [1, "1", "Yes"].

ListOfDevices Function (version 9)

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParameters](#) functions to process a group of devices.

ListOfDevices(tObjType, EString, filterName, objList1, {objList2}, {objList3})

Parameter Definitions

TobjType : String The type of object for which you are acquiring the list of devices. No default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

FilterName : String The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If the filter cannot be found, the server will default to returning all objects in the case of type TobjType.

ObjList1 : Variant This parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the first [key field](#) for the devices of type TobjType. This parameter is required for all types of devices.

ObjList2 : Variant This parameter **may** be optional. For device types that only have one [key field](#), you can omit passing a variable for this parameter. If the device type requested has two or more key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the second key field for the devices of type TobjType.

ObjList3 : Variant This parameter **may** be optional. For device types that have one or two [key fields](#), you can omit passing a variable for this parameter. If the device type requested has three key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the third key field for the devices of type TobjType.

Example

ListOfDevices("gen", EString, "", objList1, objList2)

This function call will return a list of all generators in the load flow case. By passing an empty string as the filter name, Simulator will not use an advanced filter before returning the generator key fields. Since generators have two key fields, bus number and ID, it was required to pass two variant array variables to receive the key fields. Each generator in the case would have its bus number returned in objList1 and ID in objList2. The corresponding bus number and ID are stored in the same index position in the two arrays. If only one variant array, objList1, had been passed in this instance, an error would have been returned stating more arrays were necessary. If the third variant array, objList3, had been included, Simulator would have still returned the key fields in objList1 and objList2, and would have returned objList3 as an empty variant.

LoadContingencies Function (version 9)

The LoadContingencies function can be used to read a set of predefined contingencies from a PowerWorld® Auxiliary Contingency file. Thus once you have defined a set of [contingencies](#) in Simulator and saved the list to a PowerWorld® Auxiliary file, you can specify the filename with this function and the [Simulator Automation Server](#) will load the contingencies into memory.

LoadContingencies(fileName, EString, {tAppend})

Parameter Definitions

Filename : String The name of the PowerWorld® Auxiliary file containing the defined contingencies. If the file cannot be found, an error will be returned through EString.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

tAppend : Boolean This parameter is optional. If you have more than one list of contingencies in separate files, you can append the lists together in the Simulator Automation Client by passing the Boolean value True in this parameter. The default if not passed is False, meaning any existing contingencies in the Simulator Automation Client would be cleared prior to loading the new list.

Example

LoadContingencies("c:\my files\ctgfile.aux", EString, True)

This function call will load a previously created list of contingencies that was saved in the PowerWorld Auxiliary file format. The file name string identifies the location and name of the auxiliary file. By passing the value True in the optional parameter tAppend, Simulator will read the list of contingencies and add them to any existing contingencies already loaded in Simulator. Contingencies in the list existing list with the same name as contingencies in the appending list will be replaced by the records in the appending list.

OpenCase Function (version 9)

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). If an error occurs trying to open the case, an error message will be returned through EString.

OpenCase(filename, EString)

Parameter Definitions

FileName : String The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name. If the file cannot be found or an error occurs while reading the file, an error message will be returned through EString.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

ProcessAuxFile Function (version 9)

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator. If an error occurs while processing the auxiliary file, an error message is returned through EString.

ProcessAuxFile(filename, EString)

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

RunScriptCommand Function (version 9)

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

RunScriptCommand(Statements, EString)

Parameter Definitions

Statements : String The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

Example

RunScriptCommand("Entermode(RUN); SolvePowerFlow;", EString)

This function call will switch to Run mode and then will solve the power flow of the present case.

SaveCase Function (version 9)

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file. If an error occurs while trying to save a case, an error message is returned through EString.

SaveCase(fileName, EString, {fileType}, {Overwrite})

Parameter Definitions

fileName : String The name of the file you wish to save as, including file path. No default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

fileType : String This parameter is optional. If you desire to save the case in a format other than the current PowerWorld Binary (pwb) format, you must specify the type of format in this parameter as a string. If the parameter is not passed, the format type defaults to the most recent version of PowerWorld binary (pwb) file. If a string is passed for this parameter, it must at least be the "PWB" string. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)

"PTI24" PTI version 24 (raw)

"PTI25" PTI version 25 (raw)

"PTI26" PTI version 26 (raw)

"PTI27" PTI version 27 (raw)

"GE" GE PSLF (epc)

"IEEE" IEEE common format (cf)

"PWB70" PowerWorld Binary version 7.0 (pwb)

"PWB" PowerWorld Binary (most recent) (pwb)

Overwrite : Boolean This parameter is optional. If you do not want to overwrite a file with the same name as passed in filename, you must specify this parameter as False. Default is True.

SendToExcel Function (version 9)

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed. If an error occurs while trying to send data to Excel, an error message is returned through EString.

SendToExcel(tObjectType, filterName, EString, {tFieldList})

Parameter Definitions

tObjectType : String A string describing the type of object for which you are requesting data. No Default.

filterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

tFieldList : Variant This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Example

SendToExcel("gen", "", EString, [pwBusNum, pwGenID, pwGenAGCable])

This function call will send the values of the fields in tFieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators.

WriteAuxFile Function (version 9)

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file. If an error occurs while trying to write the auxiliary file, an error message is returned through EString.

WriteAuxFile(fileName, filterName, tObjectType, EString, {tAppend}, {tFieldList})

Parameter Definitions

FileName : String The name of the PowerWorld® Auxiliary file you wish to save. No default.

filterName : String The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

tObjectType : String A string describing the type of object for which you are requesting data. No Default.

EString : Variant This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

tAppend : Boolean This parameter is optional. If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If the parameter is not passed, the Simulator Automation Server assumes false.

tFieldList : Variant This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

Example

WriteAuxFile("c:\my files\myauxfile.aux", "", "gen", EString, False, [pwBusNum, pwGenID, pwGenAGCable])

This function call will send the values of the fields in tFieldList to a PowerWorld Auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators. Since the tFieldList optional parameter was included, the tAppend optional parameter also had to be included, even though its default value is already False.

PowerWorld Object Variables (Version 9)

The ability to access power system data for different objects through various [Simulator Automation Server functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParameters](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

Examples of Field Variables

PWBusGenMW

PWBusNum

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Object Fields...** Specify a file name and location for saving the file, and click Save. Simulator will save out the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

Examples of Field Variables in Listing

PwBusGenMW Real 'Bus Gen MW'

* PwBusNum Integer 'Bus Number' (* denotes key field in text file)

ATC_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is PWBusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as PWBusPUVolt:0, and the "To" bus voltage would be represented as PWBusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.

Integrated Topology Processing Overview

The Integrated Topology Processing tool is available only if you have purchased the Integrated Topology Processing add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering Integrated Topology Processing for Simulator.

Integrated Topology Processing is a Simulator add-on, which allows you to solve EMS-type, full-topology models in a numerically robust and transparent manner. Planning cases, traditionally used in the planning environment by tools such as Simulator, correspond to a less-detailed, electrically equivalent snapshot of the full-topology model.

Integrated Topology Processing extends Simulator applications traditionally used by planners so they can operate in an operations real-time environment.

Integrated Topology Processing is fully integrated with the functions of the Simulator base package including power flow, contingency analysis, and sensitivity calculations, as well as with the [OPF](#), [SCOPF](#), [Transient Stability](#), [ATC](#), and [PVQV](#) add-ons.

Simulator's Integrated Topology Processing is different from traditional EMS topology processing, which requires converting the [full-topology model](#) into a consolidated, planning case, to numerically solve network applications such as power flow and state estimation. Simulator's Integrated Topology Processing dynamically obtains an equivalent [consolidated representation](#) of the full-topology model by dynamically reassigning the connection pointers of all of the devices. In Simulator, no "case conversion" takes place. The consolidated representation has the same topology and is electrically equivalent to the corresponding planning case.

Only the full-topology model is stored in memory. We call this novel implementation "Single-Model" Integrated Topology Processing.

Simulator's Integrated Topology Processing allows complete unification of power system planning and operations unification regarding network models and application environment. Typical uses of Integrated Topology Processing are:

- In the real-time production environment through [SimAuto](#) automation, yielding implementations of true real-time applications: real-time power flow, real-time contingency analysis (Security Assessment - SA), real-time transient stability (Dynamic Security Assessment - DSA), PVQV (Voltage Stability Assessment - VSA), real-time SCOPF (Security-Constrained Economic Dispatch - SCED).
- In near real-time operations by allowing simulation of real-time full-topology cases from the EMS.
- In the planning environment by allowing extensive study of the real-time system.

Simulator's Integrated Topology Processing consists of two main features:

- An algorithm that takes place behind the scenes that is used internally during numeric solutions such as power flow, ATC, etc.
- A tool to convert the real-time model to a traditional planning case that can be consumed by planning software not able to operate on full-topology models.

We invite you to continue reading the following Integrated Topology Processing topics:

[Full-Topology Model](#)

[Superbuses](#)

[Subnets](#)

[Consolidation](#)

[Primary Bus Priority Scheme](#)
[Illustration of Consolidation Process](#)
[Consolidated Representation](#)
[Consolidated Superbus View](#)
[Integrated Topology Processing Dialog](#)
[Power Flow Using Topology Processing](#)
[III-Conditioned Jacobian](#)
[Contingency Analysis Using Integrated Topology Processing](#)
[Incremental Topology Processing](#)
[Integrated Topology Processing in Applications](#)
[Saving the Planning Case](#)
[Planning Cases](#)
[Primary Bus Mapping](#)
[Breaker-to-Device Contingency Conversion](#)
[Device Derived Status](#)

Integrated Topology Processing: Full-Topology Model

Real-time applications require modeling the power network at a level of detail that includes all the switching devices such as circuit breakers and disconnects and other very low-impedance branch devices. For instance, an operator must have information about specific breaker statuses to coordinate device maintenance procedures.

These detailed models are referred to as “node-breaker” or “full-topology” models. This type of model cannot be solved directly in conventional planning applications because of the large number of very low-impedance branches resulting from the switching devices. The power flow would be forced to use an [Ill-Conditioned Jacobian](#) matrix if these low-impedances were directly modeled. In order to solve the power flow on a full-topology model, Simulator dynamically obtains a [Consolidated Representation](#) of the power network by dynamically moving the device connection pointers.

Simulator models all type of branch objects as transmission line records. To aid in the consolidation, additional input parameters are necessary to properly identify the type of branch object that is being modeled and determine whether or not that branch can be eliminated during consolidation. These fields are accessible from the Topology group in the Available Fields for the [Line and Transformer Display](#) in the [Model Explorer](#). Two relevant input fields needed for Topology Processing are:

Branch Device Type

Type of branch specified as *Transformer*, *Line*, *SeriesCap*, *Breaker*, *Load Break Disconnect*, *Disconnect*, *ZBR* (Zero-Impedance Branch), *Fuse*, and *Ground Disconnect*.

For brevity in the context of tools that either automatically switch or identify breakers to either open or close a device, the term *breaker* is used to indicate any type of device that can be switched automatically. This includes branch devices of type *Breaker* and *Load Break Disconnect*.

Allow Consolidation of Branch

This is a *YES* or *NO* field, which tells Simulator to eliminate closed switching devices and other very low-impedance branches from the network model during consolidation. Only branches that have a **Branch Device Type** equal to *Breaker*, *Load Break Disconnect*, *Disconnect*, *ZBR*, *Fuse*, or *Ground Disconnect* can be consolidated. If an element is of this type and is set not to be consolidated, it will be preserved in the model and its default impedance value will be used in the power flow equations. In the context of consolidation, the term *breaker or switching device* can be used to include any of the device types that can be consolidated.

There are other instances in which a branch that is marked as allowed to be consolidated will not be consolidated:

- Branches that are area tie lines
- Branches that are connected to a multi-terminal dc line converter
- Branches that are part of a [Model Condition](#)
- Branches that are part of a [Model Expression](#)
- Branches that are part of a [Post Power Flow Solution action](#)
- Branches that are part of [Transient Stability event](#)
- Branches that are part of a contingency action if not using [Incremental Topology Processing](#)
- Branches that are part of an interface unless the branch is in series with a device that cannot be consolidated, e.g. transformer, line, load, generator, etc. The interface will be automatically modified to monitor the series device instead of the branch so that the branch can be consolidated.

Simulator models all of the system connection points: Busbars, Junctions, Terminals, etc. as buses.

Topology Bus Type

Type of connection point. Valid entries include *BusbarSection*, *Junction*, *Internal_3WND*, *Ground*.

Topology Processing provides an optional input field for buses to aid in the consolidation:

Node Priority

Priority level of the bus to become a primary bus. This overrides the default Primary Bus Priority Scheme level for this bus.

Topology Processing determines two bus fields as part of the consolidation:

Primary Node

Bus to which all the device connection pointers in a Superbus will point to during Consolidation.

Node Neighbor List

If this is a Primary Bus, this contains a list of all buses that belong to the superbus in which this bus is contained. This entry is blank for buses that are not primary buses.

These four bus fields are available under the Topology group in the Available Fields for the [Bus Display](#) found on the [Model Explorer](#).

Integrated Topology Processing: Superbuses

A SuperBus in the [Full-Topology Model](#) is a group of buses connected through CLOSED switching devices or other CLOSED very low-impedance branch devices. All the buses in a Superbus have the same voltage phasor, i.e., they correspond to the same electric point. As the status of switching devices change, superbuses change correspondingly, sometimes resulting in superbus merging or splitting.

Superbuses are set automatically when the case is read from the binary or aux file and they are updated at the beginning of a power flow solution or sensitivity calculation to capture possible changes in the status of switching devices. The details of the Superbus records is available in the [Model Explorer](#) under **Solution Details > Superbuses**.

A superbus in the [Full-Topology Model](#) would correspond to a bus in the electrically equivalent bus/branch planning case.

Each Superbus in a system belongs to only one [Subnet](#).

Integrated Topology Processing: Subnets

A Subnet is a group of buses connected through switching devices or other very low-impedance branches devices regardless of their open or closed status. Subnets are bounded by transmission line and transformer terminals. Subnets on a system hence do not depend on the status of breakers, but only on the presence of devices. Subnets do not change unless equipment is added or removed from the physical model.

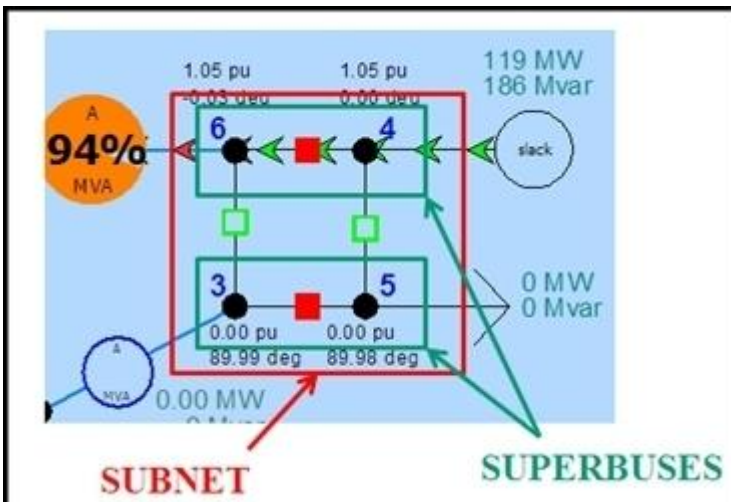
A Subnet can contain one or more [Superbuses](#).

A [Superbus](#) belongs to only one subnet.

Subnets and Superbuses are set automatically when the case is read from the binary or aux file.

The list of subnets and the details of subnet records can be seen on the [Model Explorer](#) under **Solution Details > Subnets**.

The example below shows a subnet that has four nodes. The shown breakers statuses results in two superbuses. If either breaker is closed, the two superbuses will merge.



Integrated Topology Processing: Consolidation

Consolidation consists in moving the connection pointers of all the devices within a Superbus to a specially selected bus called the Primary Bus, and hiding not needed switching devices and buses. The Primary Bus is selected based on a default [Primary Bus Priority Scheme](#).

Prior to consolidation, all the power system devices in a superbus, such as generators, loads, shunts and transmission line terminals are connected to their actual terminal buses in the physical system. After consolidation all the devices are connected to their corresponding Primary Bus. This makes the status and actually, the presence of switching devices in a superbus irrelevant, allowing the application to disregard them during the numerical solutions. A list of the branch device types that are allowed to be consolidated and exceptions to when these branch device types are not consolidated can be found in the [Full-Topology Model](#) topic. We say that the full-topology model is in a consolidated state and we call the representation of the system after consolidation, the [Consolidated Representation](#).

After consolidation, the topology of the full-topology model is identical to that of an equivalent planning case and the models are electrically identical. However, in Simulator no case conversion takes place. Only the full-topology model exists in memory.

The following link contains an example illustrating the consolidation process: [Illustration of Process](#).

The user is never allowed to interact with the model while it is in a consolidated state.

The user does not have access to the Consolidated Representation unless it is saved as a planning case as described in the section [Saving the Planning Case](#).

Integrated Topology Processing Consolidation: Primary Bus Priority Scheme

During [Consolidation](#), the devices connected to the buses of a [superbus](#) change their connection pointers to point to the Primary Bus. In this manner, all the devices appear to be connected to primary buses only.

By default, the selection of the primary bus takes place using the following priority scheme, in descending order of priority:

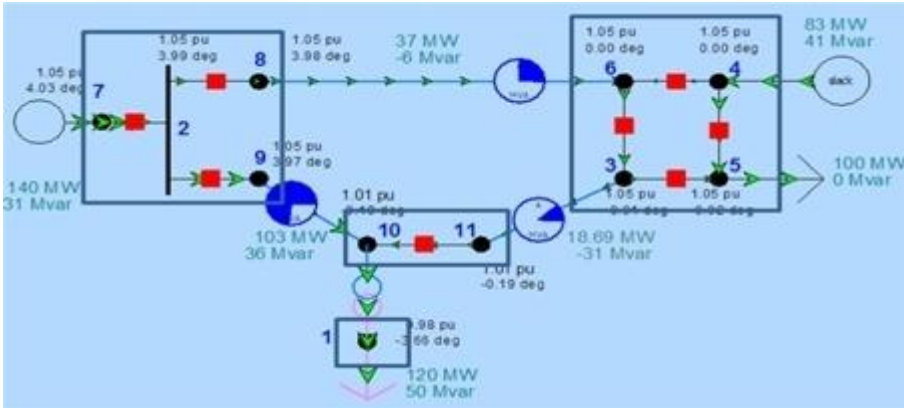
1. Slack bus (highest priority)
2. Multi-Terminal DC Line Terminals
3. Generator Regulated Bus
4. Switched Shunt Regulated Bus
5. LTC Regulated Bus
6. DC Line Terminal
7. Generator Terminal
8. Switched Shunt Terminal
9. Load Terminal
10. Branch with Device Type = Series Capacitor
11. Branch with Device Type = Transformer
12. Branch with Device Type = Line
13. Branch with Device Type = ZBR
14. Branch with Device Type = Breaker
15. Branch with Device Type = Load Break Disconnect
16. Branch with Device Type = Disconnect
17. Branch with Device Type = Fuse
18. Branch with Device Type = Ground Disconnect

In case of ties, the minimum bus number is selected as the primary bus.

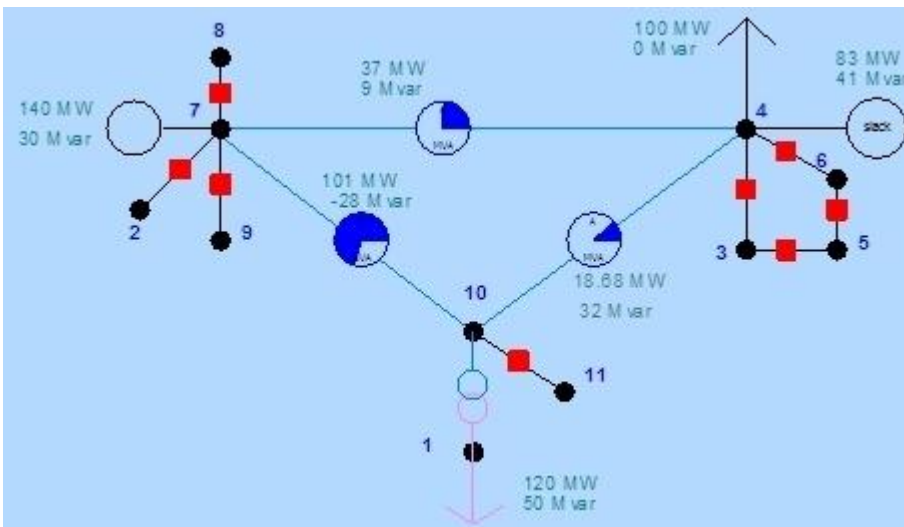
There is also an optional integer field available with each bus called **Topology\Node Priority**. When this is specified for a bus, the bus with the highest Node Priority is chosen as the primary node. By default this Node Priority is 0 for all buses, and thus the priority defaults to the scheme specified above.

Integrated Topology Processing Consolidation: Illustration of Process

Consider the following 11-bus, [full-topology model](#). Given the breaker statuses, this system would contain 4 [superbuses](#) shown in the next figure.



We apply [consolidation](#) to the system above considering that the primary buses of each superbuse are buses 1, 4, 7 and 10. The system topology at this point (which is never seen by the user) would look as follows.



At this consolidated state, the breakers and the non primary bus buses could be removed without affecting the power flow solution.

Integrated Topology Processing: Consolidated Representation

A consolidated representation of a full-topology model is one obtained when the system is in a consolidated state. In this state all the connection pointers of devices and device terminals have been moved to the primary buses of the [Superbuses](#) as part of the [Consolidation](#) process.

The user does not have access to the Consolidated Representation records unless it is saved as a planning case as described in the section [Saving the Planning Case](#). However, the Consolidated Representation can be navigated using the [Consolidated Superbus View](#).

Integrated Topology Processing: Consolidated Superbus View

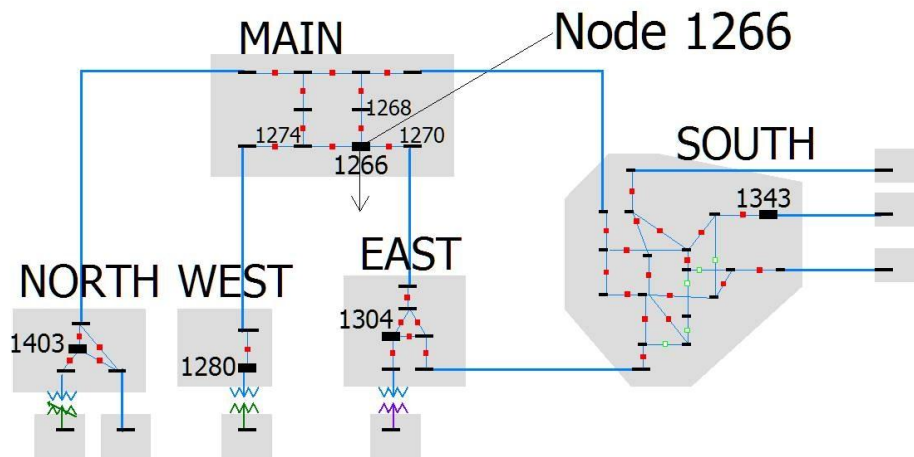
Simulator's Bus View provides a feature that allows a [Full-Topology Model](#) to be navigated using the [Consolidated Representation](#), in which only the primary buses are shown. This gives the appearance to the user of navigating the electrically equivalent planning case.

The Consolidated Superbus View is accessed from the [Bus View Display](#). If the current case is set up such that [superbuses](#) can be identified, there will be a drop-down box at the top of the Bus View Display that will allow switching between the Full Topology representation and the Consolidated Superbus representation.

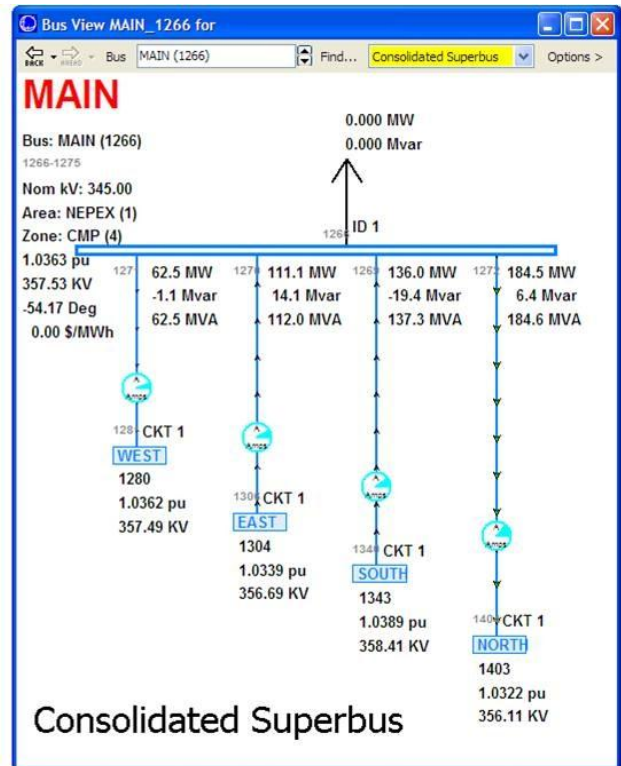
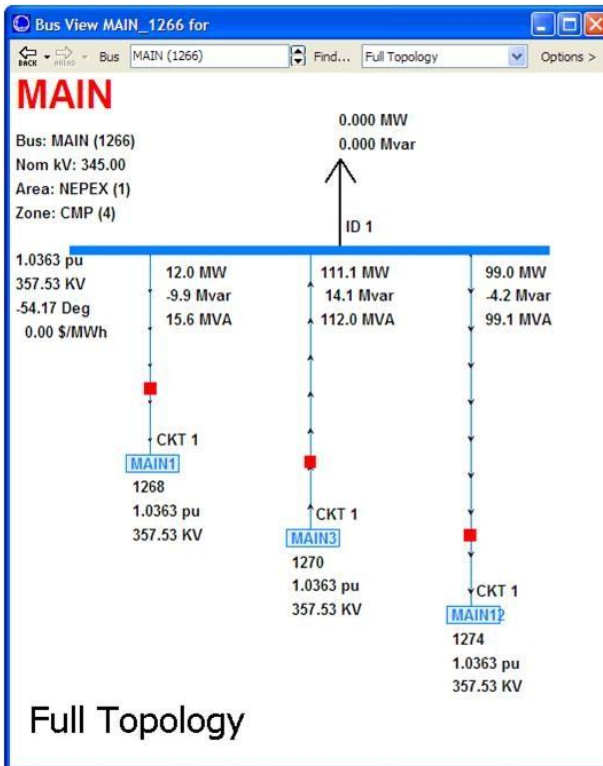
The Consolidated Superbus View has the following conventions:

- Shows radial-connected generator, load, and switched shunt buses above the main (primary) bus
- For series connections of buses, the main branch shown will be the first branch that is marked as NOT available for consolidation (i.e. AllowConsolidation = *NO*)
- All generator, load, and switched shunt devices will appear connected to their Primary Bus instead of the actual bus
- Only branches that connect different superbuses will be shown. This means that closed circuit breaker will not appear on the consolidated superbuses view because they connect two buses that are in the same superbus.

An example full-topology model is shown below. Each gray region represents a superbus. The large buses and numbers in each superbus represent the primary bus for that superbus.

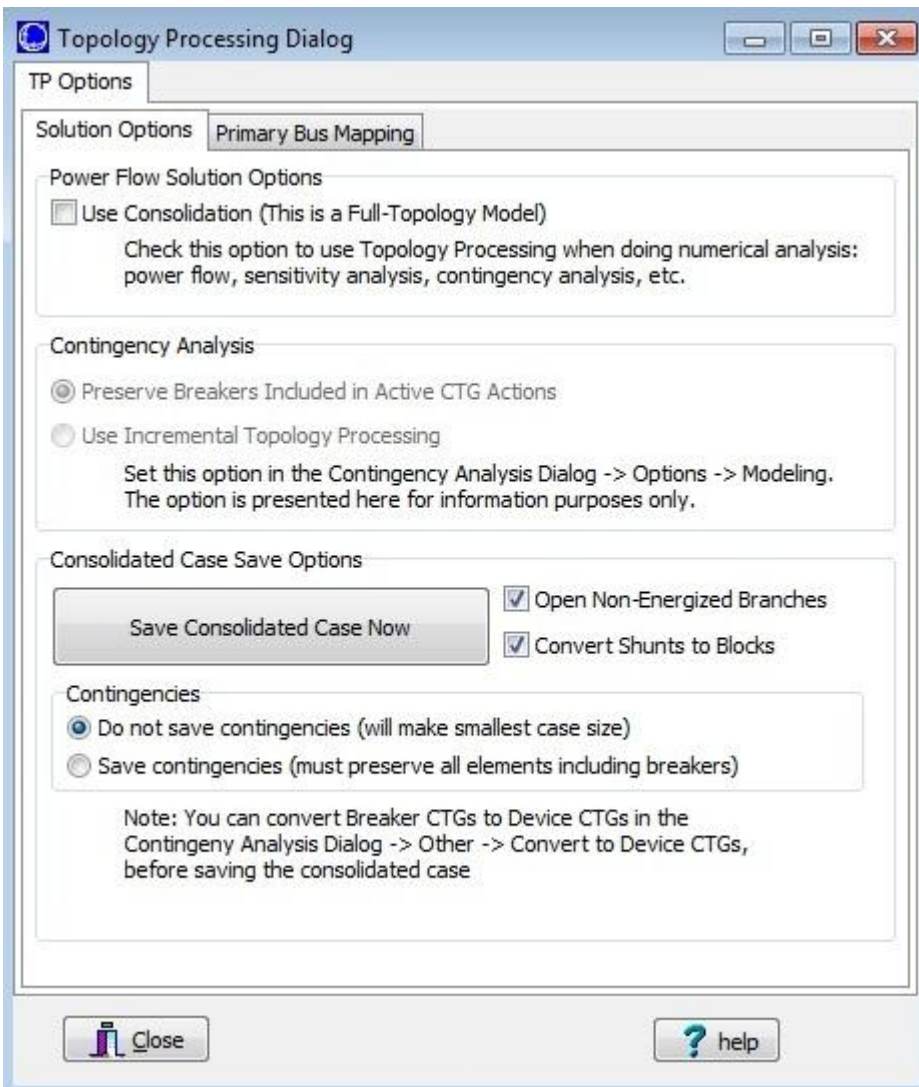


The associated full-topology bus view and the consolidated superbuses view for the above model is shown below:



Integrated Topology Processing Dialog

To open the Topology Processing Dialog select **Topology Processing** found on the [Add Ons](#) ribbon tab in the **Topology Processing** ribbon group. This dialog includes options to solve the [Power Flow Using Topology Processing](#), perform [Contingency Analysis Using Topology Processing](#), and for [Saving the Consolidated Case](#).



Power Flow Solution Options

Use Consolidation (This is a Full-Topology Model)

This option is used for all Simulator [applications](#): power flow, contingency analysis, sensitivity calculations, etc. This option should ALWAYS be checked for real-time cases, while it should be unchecked for planning cases. If a full-topology model was read and this option is not checked, the power flow will try to solve the model without removing the switching devices resulting in a very large [ill-conditioned Jacobian](#). This option is also found on the [Power Flow Solution Common Options](#) page of the [Simulator Options](#) dialog.

Contingency Analysis

These options are shown on this dialog for informational purposes only. These can be accessed and set from the [Contingency Analysis dialog](#) on the [Options](#) tab and [Modeling](#) sub-tab. These options are described in more detail in the [Contingency Analysis Using Topology Processing](#) topic.

Consolidated Case Save Options

These options allow the saving of the consolidated representation as a traditional planning case. More detail about these options is provided in the [Saving the Planning Case](#) topic.

Options found on the **Primary Bus Mapping** tab are described in the [Primary Bus Mapping](#) topic.

Integrated Topology Processing: Power Flow Using Topology Processing

In order to perform numerical solutions using Integrated Topology Processing, Simulator does the following:

[Consolidates](#) the system relocating the devices and removing not needed breakers and non primary buses.

Performs the numerical calculation.

De-consolidates the system by restoring breakers and relocating the devices to their original buses. This step also maps the values of the primary buses to all the buses in the [superbus](#).

This is transparent to the user, who only needs to tell Simulator to use consolidation. The option that needs to be set to use consolidation is found on the [Topology Processing Dialog](#) and also on the [Power Flow Solution](#) page of the [Simulator Options](#) dialog.

Integrated Topology Processing: Ill-Conditioned Jacobian

Matrix **A** in the linear equation $\mathbf{Ax} = \mathbf{b}$ is ill-conditioned if small changes in the entries of the vector **b** radically change the solution vector **x**. Ill-conditioned matrices are particularly troublesome in iterative solvers such as AC power flow algorithm implementations. One iteration of the AC power flow solves the linear equations $\mathbf{J} \Delta \mathbf{x} = \Delta \mathbf{y}$ where **J** is the Jacobian matrix, $\Delta \mathbf{x}$ is the vector of voltage magnitude and angle differences and $\Delta \mathbf{y}$ is the vector of real and reactive power mismatches.

A power flow Jacobian can become ill-conditioned if the system branches present reactances that differ in orders of magnitude, such as very short lines or zero impedance branches. Because the reactance of switching devices is negligible compared to that of transmission lines and transformers, modeling but a short number of switching devices in the Jacobian is prohibitive. The lower the value of the reactance, the more ill-conditioned the Jacobian will become. On the other hand, if the assumed reactance value is not too low, the solution would be inaccurate resulting in larger bus voltage angles. In Simulator switching devices that are present in [Full-Topology Models](#) are handled by [Integrated Topology Processing](#), producing an exact and robust numerical solution.

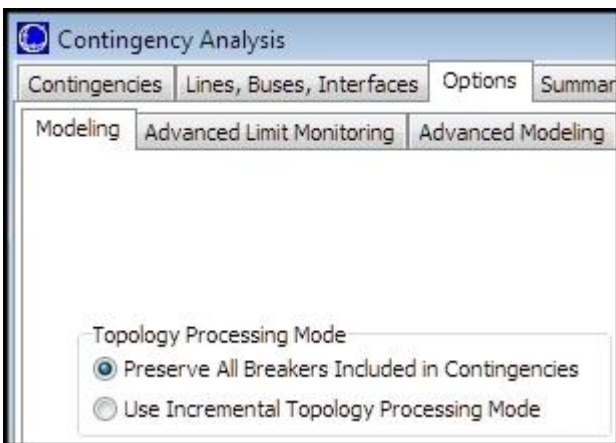
Integrated Topology Processing: Contingency Analysis Using Topology Processing

Simulator Integrated Topology Processing allows solving contingencies in the [Full-Topology Model](#).

EMS systems model contingencies through a list of real system actions. For instance, opening a line involves opening the set of breakers that electrically isolate the device. The same contingency action in a planning environment would be modeled as a single contingency action: OPEN LINE. Simulator provides a special contingency action called [Open with Breakers](#) which enables this type of modeling. There is also a [Close with Breakers](#) contingency action that enables modeling breakers to energize a device instead of using a planning-type contingency action to change the status of the device itself.

There is an option to determine whether to monitor all buses or only the primary node in each superbuses during contingency analysis, **When using Integrated Topology Processing, monitor only the primary bus for each superbuses**. This is found on the [Contingency Analysis dialog](#) on the [Options](#) tab on the [Limit Monitoring](#) page.

The option to determine how contingencies should be modeled within topology processing is found on the [Contingency Analysis dialog](#) on the [Options](#) tab and [Modeling\Basics](#) page in the **Topology Processing Mode** grouping.



Preserve All Breakers Included in Contingencies

If this option is selected, contingency analysis will not treat contingency breakers as zero impedance branches and will keep all of them in the model. Hence, the consolidated representation is obtained only once upfront before the first contingency, and this representation is used to solve all the contingencies. Using this option, the consolidated representation is smaller than the full model, but is not the smallest possible. If the number of contingencies and breaker actions is large, many breakers will be preserved in the case. If the percentage of preserved breakers is significant, this may result in an [Ill-Conditioned Jacobian](#) during the contingency solution.

Use Incremental Topology Processing Mode

In this option, contingency analysis uses an [Incremental Topology Processing](#) approach to solve each contingency. Since usually the number of contingency actions in a contingency is small (usually around 4 to 10), only a small number of [Subnets](#) are affected by a contingency. The topology of all the other subnets remains unchanged. When this option is used, Simulator does a full consolidation before the first contingency and then for each contingency, the topology of just the affected subnets is re-processed (determining superbuses and primary buses, and updating the connection pointers).

Incremental Topology Processing provides a more robust solution and should be used whenever possible.

Incremental Topology Processing cannot be used for applications that use linearization and that require a constant size of the bus array, such as sensitivity analysis. For instance, ATC will automatically use the contingency analysis **Preserve Breakers Included in Contingencies** option.

Contingency Element: Open Breakers

The contingency action type *Open Breakers* is available for each contingency element type for which the *Open* action type is available. This includes the following element types: Branch, Bus, Load, Generator, Switched Shunt, DC Line, Injection Group, and Interface. This action type is only available for contingency records and is not available for [contingency block definitions](#) or for [global contingency actions](#).

In order to handle the *Open Breakers* contingency elements, before one contingency's solution is performed, the list of contingency elements for that contingency is processed by looking for elements which use *Open Breakers*. For elements with this action type, all the terminal buses of the respective device are processed using the algorithm below to determine which breakers to open to isolate the device (and possible which generators to outage). The basic idea of the algorithm is to search for a list of breakers that will isolate all the terminal buses of the device from in-service generation. The algorithm will also return a list of online generators which must be opened to isolate the bus as well. Remember that a breaker is specified by setting the **Branch Device Type** to *Breaker*.

Using the list of breakers and generators found from the algorithm, contingency analysis augments the contingency element list with a dynamically created list of contingency elements of action type *Open* for each breaker and generator device. The dynamically created elements are given the same [Model Criteria and Status](#) as the contingency element which required their creation. Note that if multiple *Open Breakers* contingency elements require the same breaker to open, then one new contingency element will be created. When *Open Breakers* contingency elements have different Model Criteria, then a new contingency element will be created for each unique Model Criteria. During the actual contingency solution, the *What Actually Occurred?* list will be updated and can be examined in the results to determine what breakers were needed to isolate the device. Finally, after the contingency solution is completed, all the dynamically created contingency elements are deleted, and as a result the user can only see which breakers or generators were opened by looking at the *What Actually Occurred* list.

For brevity in the context of tools that either automatically switch or identify breakers to close a device, the term *breaker* is used to indicate any type of device that can be switched automatically. This includes **Branch Device Types** *Breaker* and *Load Break Disconnect*. If the model has no branches defined with either of these Branch Device Types, this option will not work because the algorithm will never find any breakers. See the [Full Topology Model](#) topic for more information on defining breakers.

Algorithm description for Open with Breakers

This algorithm is applied to each terminal bus of the device specified. For generators, loads, shunt, and buses this is done on a single bus only. For an Injection Group however, each generator, load, and shunt in the injection group will be processed in turn. Starting at each terminal bus of the device and marching outwards, the algorithm traverses closed branches (except for breakers), dc lines, and multi-terminal DC lines and will flag any closed breakers at buses which are visited. The buses that are visited are also flagged. (Note: the algorithm will NOT traverse across the breakers with for an exception noted below). When buses are visited that have online generators, a list of generators is maintained that must be opened as well. In order to prevent the degenerate case where breakers do not fully isolate any portion of the network, only 10 buses with online generation are permitted for visitation. If more than 10 buses are encountered with online generation, then the algorithm will immediately abort and a log message is written saying that the device "cannot be isolated from online generation".

If this part of the algorithm completes it will have obtained a list of flagged breakers and a list of visited buses (as well as a list of generators to open). The list of flagged breakers is then examined to ensure that each breaker actually isolates the original device. Each flagged breaker will be examined and if a flagged breaker has a terminal bus that is NOT in the list of visited buses, then

this breaker disconnects the device from another part of the system and will be maintained. Breakers for which both terminal buses have been visited will be discarded because these breakers do not actually isolate anything. If at least one flagged breaker is maintained then the process is deemed successful and the contingency will outage the list of maintained flagged breakers. If all the flagged breakers are discarded, then the breakers do not isolate anything, and therefore a log message is written saying that "no part of the system can be isolated," and nothing is done.

A few special notes:

1. If there is another contingency action inside this same contingency that specifies an action type of *Open* or *OpenCBs* for a generator device, encountering that generator won't count toward the list of generators to open or the maximum of 10 buses to encounter with online generation. The same is done for any generator connected to a bus or inside an injection group, if the bus or injection group has a contingency action of type *Open* or *OpenCBs*.
2. If there is a contingency action inside this same contingency that specifies a breaker with action type *Open Breakers*, the algorithm will traverse across this breaker (this models a breaker failure where a breaker is supposed to open but fails to operate.)
3. If there is a contingency action inside this same contingency which specifies an action type of *Open* for a branch or DC line device, the algorithm will NOT traverse across that branch or DC line. The same is done for any branch which is part of a three-winding transformer, interface, or connected to a bus if the three-winding transformer, interface, or bus has a contingency action of type *Open*.
4. While a list of generators to open is maintained, if no online load is encountered while traversing buses in the algorithm, then the generator OPEN contingency actions will not be created. This is because the breakers will create an isolated island with no load and thus opening the generation is not necessary.

Integrated Topology Processing: Incremental Topology Processing

Incremental Topology Processing is a technique used to update the [Consolidated Representation](#) of the system in a very fast manner. It is particularly useful during [Contingency Analysis](#), where it does the following:

- Consolidate entire case

- For each contingency do

- Apply switching device actions and identify subnets with changed topology

- Expand affected subnets to full-topology detail

- Reconsolidate affected subnets with post-contingency topology

- Solve post-contingency power flow

- Determine and store limit violations

- Reset affected subnets to pre-contingency topology

- Restore pre-contingency power system state

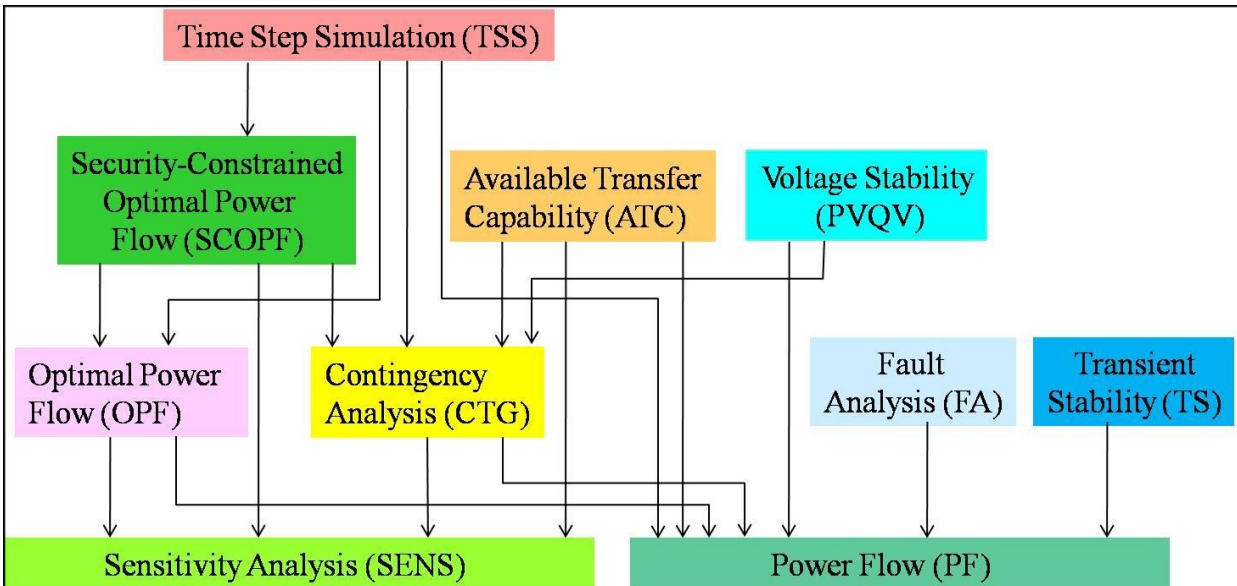
- End

- Deconsolidate entire case

Incremental Topology Processing is very fast, taking on the order of a few milliseconds.

Integrated Topology Processing: Topology Processing in Applications

Simulator applications such as ATC and OPF use a hierarchical scheme where [Consolidation](#) takes place only at the beginning of the numerical solution process. Consolidation in lower functions takes place only if it has not taken place already in higher-level functions.



Following completion of the application, de-consolidation occurs in the function level in which the consolidation originally occurred. This process is very similar to the one described for [solving the power flow using integrated topology processing](#). There can be some exceptions to this when using Contingency Analysis. The [Contingency Analysis Using Integrated Topology Processing](#) topic provides more details on this.

Once the input data is set in the [Full-Topology Model](#) fields, the applications work in the same manner as with regular [Planning Cases](#).

Integrated Topology Processing: Saving the Planning Case

Integrated Topology Processing allows you to save the [Consolidated Representation](#) of the [Full-Topology Model](#) as a traditional [Planning Case](#) (bus/branch model), in any of the [formats](#) supported by Simulator.

Although the saved case can be reopened in Simulator as a regular case, the user should note that the topology of this case will depend on the statuses of the breakers in the Full-Topology Model. If the statuses of the switching devices in the Full-Topology Model are continuously changing as is the case when the case comes from a real-time Retriever implementation, then the topology of the saved cases will be slightly different minute to minute.

The saving tool is available on the [Topology Processing Dialog](#) in the Consolidated Case Save Options section.

The screenshot shows a dialog box titled "Consolidated Case Save Options". It contains a button labeled "Save Consolidated Case Now" which is highlighted with a dashed blue border. To the right of the button are two checked checkboxes: "Open Non-Energized Branches" and "Convert Shunts to Blocks". Below these is a section titled "Contingencies" with two radio button options: "Do not save contingencies (will make smallest case size)" (which is selected) and "Save contingencies (must preserve all elements including breakers)". At the bottom of the dialog, there is a note: "Note: You can convert Breaker CTGs to Device CTGs in the Contingency Analysis Dialog -> Other -> Convert to Device CTGs, before saving the consolidated case".

Save Consolidated Case Now

Click this button to save the case in any of the supported [formats](#).

Open Non-Energized Branches

If the terminal buses of a transmission line or transformer are not energized, then assume the element is open.

Convert Shunts to Blocks

Some systems have several shunts in a superbus, and some formats support only one shunt per bus. This option allows modeling those shunts as an aggregate device.

Do not save contingencies (will make smallest case size)

Use this option if contingency analysis will not take place on the saved case.

Save contingencies (must preserve all elements including breakers)

All the contingencies, the contingency actions, and all the switching devices included in contingency actions will be preserved and saved in the consolidated case. It makes a big difference to perform a [Breaker-to-Device Contingency Conversion](#) before saving the case. Usually most of the breaker contingencies can be converted to device contingencies requiring just a small number of breakers to be preserved in the case. Some planning tools have limitations regarding the number of low impedance branches that can be modeled and will not be able to read and solve the [Planning Case](#) if it contains a large number of switching devices.

Integrated Topology Processing: Planning Cases

Planning Cases are a high-level, simplified view of the actual network, in which all the switching devices have been removed. Planning Cases use a bus/branch model that uses the bus numbers as key fields for all the system elements. [Full-Topology Models](#) on the other hand, are detailed representations of the physical system that include circuit breakers and disconnects. Simulator Topology Processing allows complete model unification and provides a unified application environment to simulate both types of models.

Integrated Topology Processing: Saving the Planning Case

Integrated Topology Processing allows you to save the [Consolidated Representation](#) of the [Full-Topology Model](#) as a traditional [Planning Case](#) (bus/branch model), in any of the [formats](#) supported by Simulator.

Although the saved case can be reopened in Simulator as a regular case, the user should note that the topology of this case will depend on the statuses of the breakers in the Full-Topology Model. If the statuses of the switching devices in the Full-Topology Model are continuously changing as is the case when the case comes from a real-time Retriever implementation, then the topology of the saved cases will be slightly different minute to minute.

The saving tool is available on the [Topology Processing Dialog](#) in the Consolidated Case Save Options section.

The screenshot shows a dialog box titled "Consolidated Case Save Options". It contains a button labeled "Save Consolidated Case Now" which is highlighted with a dashed border. To the right of the button are two checked checkboxes: "Open Non-Energized Branches" and "Convert Shunts to Blocks". Below these is a section titled "Contingencies" with two radio button options: "Do not save contingencies (will make smallest case size)" (which is selected) and "Save contingencies (must preserve all elements including breakers)". At the bottom of the dialog, there is a note: "Note: You can convert Breaker CTGs to Device CTGs in the Contingency Analysis Dialog -> Other -> Convert to Device CTGs, before saving the consolidated case".

Save Consolidated Case Now

Click this button to save the case in any of the supported [formats](#).

Open Non-Energized Branches

If the terminal buses of a transmission line or transformer are not energized, then assume the element is open.

Convert Shunts to Blocks

Some systems have several shunts in a superbus, and some formats support only one shunt per bus. This option allows modeling those shunts as an aggregate device.

Do not save contingencies (will make smallest case size)

Use this option if contingency analysis will not take place on the saved case.

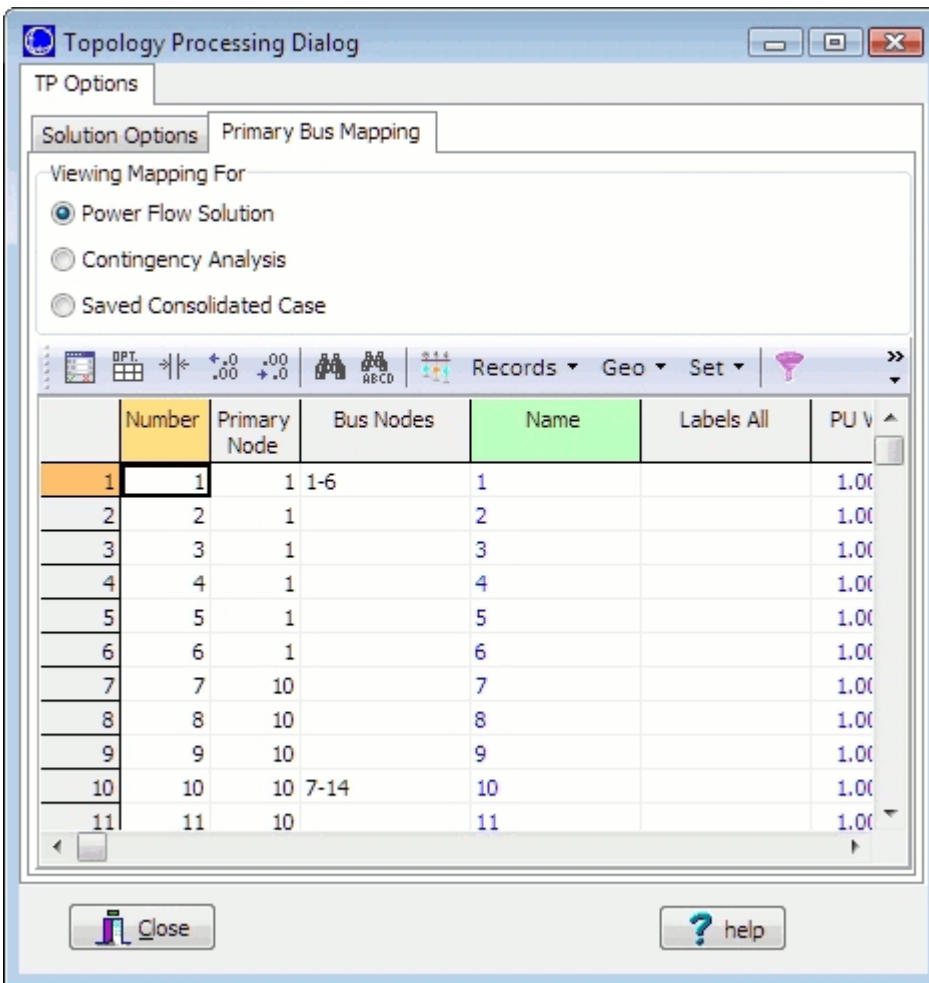
Save contingencies (must preserve all elements including breakers)

All the contingencies, the contingency actions, and all the switching devices included in contingency actions will be preserved and saved in the consolidated case. It makes a big difference to perform a [Breaker-to-Device Contingency Conversion](#) before saving the case. Usually most of the breaker contingencies can be converted to device contingencies requiring just a small number of breakers to be preserved in the case. Some planning tools have limitations regarding the number of low impedance branches that can be modeled and will not be able to read and solve the [Planning Case](#) if it contains a large number of switching devices.

Integrated Topology Processing: Primary Bus Mapping

Sometimes, it is necessary to explore or export the topology relations between [superbuses](#) and buses in the [Full-Topology Model](#) and its [Consolidated Representation](#). Because different sets of switching devices can be preserved by Simulator when solving the [Power Flow Using Topology Processing](#), when solving the [Contingency Analysis Using Topology Processing](#), and when [Saving the Planning Case](#), there will be different mappings for each one of these actions.

Primary Bus Mapping options are available on the **Primary Bus Mapping** tab of the [Topology Processing Dialog](#).



This display shows a detailed mapping of superbuses to buses. By changing the selection for which mapping to view, the corresponding mapping that would take place is shown in this [case information display](#).

Power Flow Solution

Provides the mapping used when solving the power flow.

Contingency Analysis

Provides the mapping used when solving contingencies.

Saved Consolidated Case

Provides the corresponding topology relations between the full-topology model and the consolidated case to be saved.

This mapping can be saved as an auxiliary file from the local menu of the display (right-click and choose **Save As > Auxiliary File...**) or from the [case information toolbar](#).

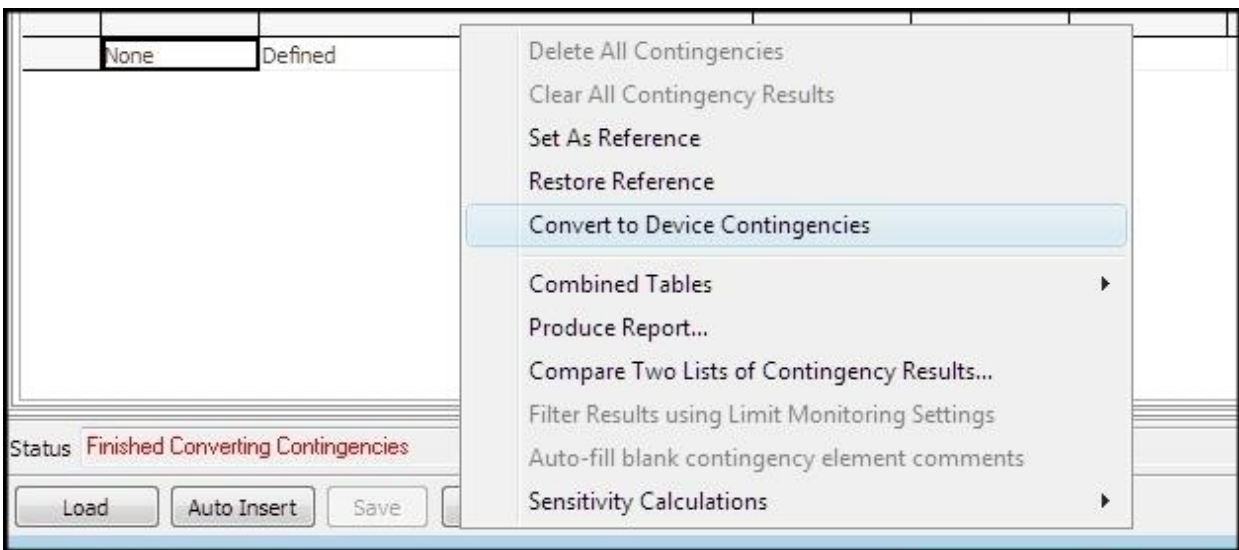
Integrated Topology Processing: Breaker-to-Device Contingency Conversion

While operation contingencies are defined using breaker actions, planning cases use simplified device [contingency actions](#). For instance, the outage of a single transmission line is modeled in EMS systems as the set of the actual circuit breaker actions that physically de-energize the transmission line. On the planning side, the same contingency is modeled as a single contingency action related to the transmission device: OPEN LINE.

A feature of PowerWorld's [Integrated Topology Processing](#) is the ability to convert operations-type, circuit breaker contingencies to their electrically equivalent device contingencies for single outage, multiple outages, or complex post-contingency topologies. While both the breaker contingencies and the device contingencies can be solved on the [Full-Topology Model](#), breaker contingencies **MUST** be converted to device contingencies in order to solve them on the consolidated [Planning Case](#).

Some complex breaker contingencies cannot be converted to electrically equivalent topologies by simply changing the energization status of devices. A subset of the breakers involved in those contingencies may need to be preserved in the case to entirely reproduce the electric topology.

The Breaker-to-Device Contingency Conversion is launched from the [Contingency Analysis Dialog](#) > **Other** button > **Convert to Device Contingencies**. This routine will convert all the contingencies in the list. The contingency analysis status box will indicate the progress of the contingency conversion process. The log will also provide messages during the conversion.



The new list of converted device contingencies will replace the original list of breaker contingencies previously displayed on the [Contingency Analysis Dialog](#). The list of device contingencies can be solved directly on the [Full-Topology Model](#). On average, device contingencies include less device actions per contingency compared to breaker actions. For instance, a single line outage usually requires opening several circuit breakers, whereas the device contingency equivalent requires a single OPEN LINE contingency action.

In order to reflect the conversion process, the actions in the [Contingency Definitions](#) include device actions with the name of the device: *Line*, *Transformer*, *Load*, *Gen*, etc. and also *Breaker* if breakers need to be preserved for a given contingency.

Device Derived Status

Differences exist between planning models and full-topology EMS models when it comes to determining if a device is energized and what the status is of the device.

With planning software and bus-branch models, two distinct fields exist: **Status** and **Energized**. The **Status** field is an explicit field that exists to determine if a device is open or closed. This is needed because breakers are not modeled. The **Energized** field indicates if the device is actually energized which will be affected by the status of branches.

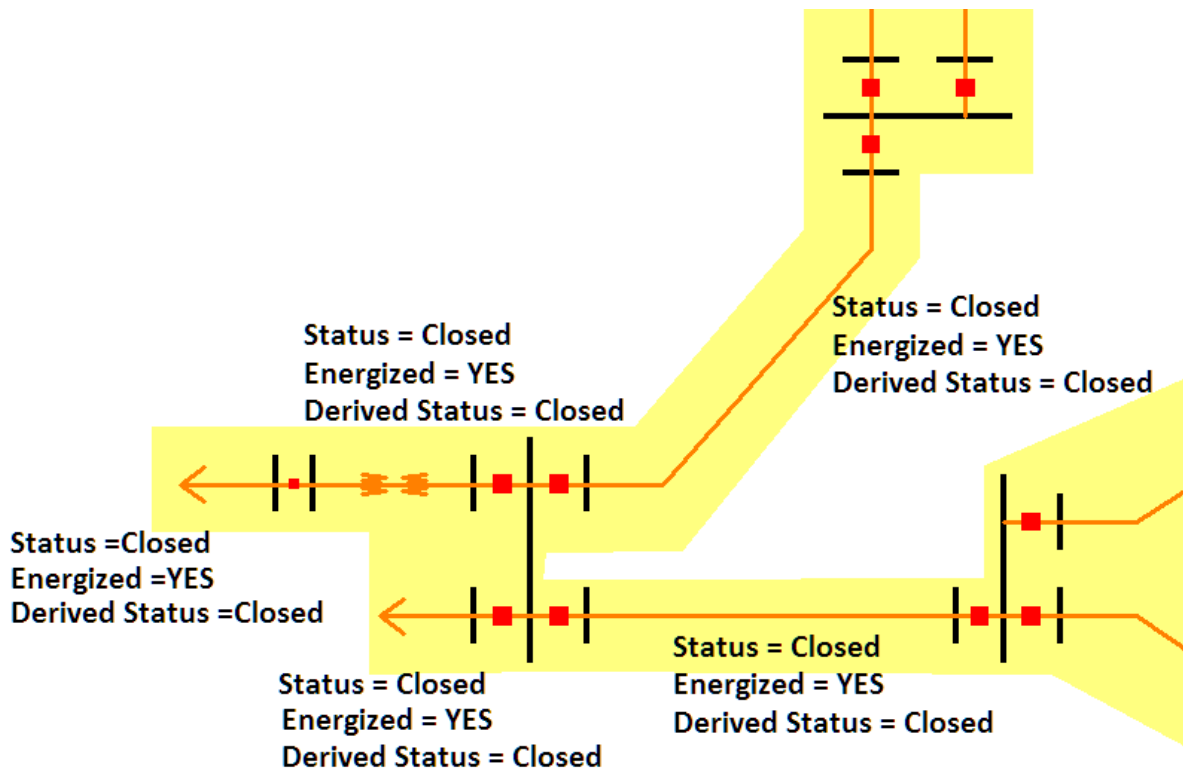
With full-topology EMS models, breaker or disconnect status determines the status of other devices, i.e. whether or not they are energized. No explicit status field exists for other devices like generators, loads, lines, transformers, etc. Typically when using a full-topology model in Simulator, the **Status** of non-switching devices (generators, loads, lines, etc.) will be set to *Closed*. A hybrid model with only parts of the system modeled with breaker detail can still use the **Status** field for non-switching devices.

The actual status of a device can be confusing, especially for those accustomed to a planning model. To eliminate confusion, the **Derived Status** field will be used to indicate the status with which planning model users are accustomed.

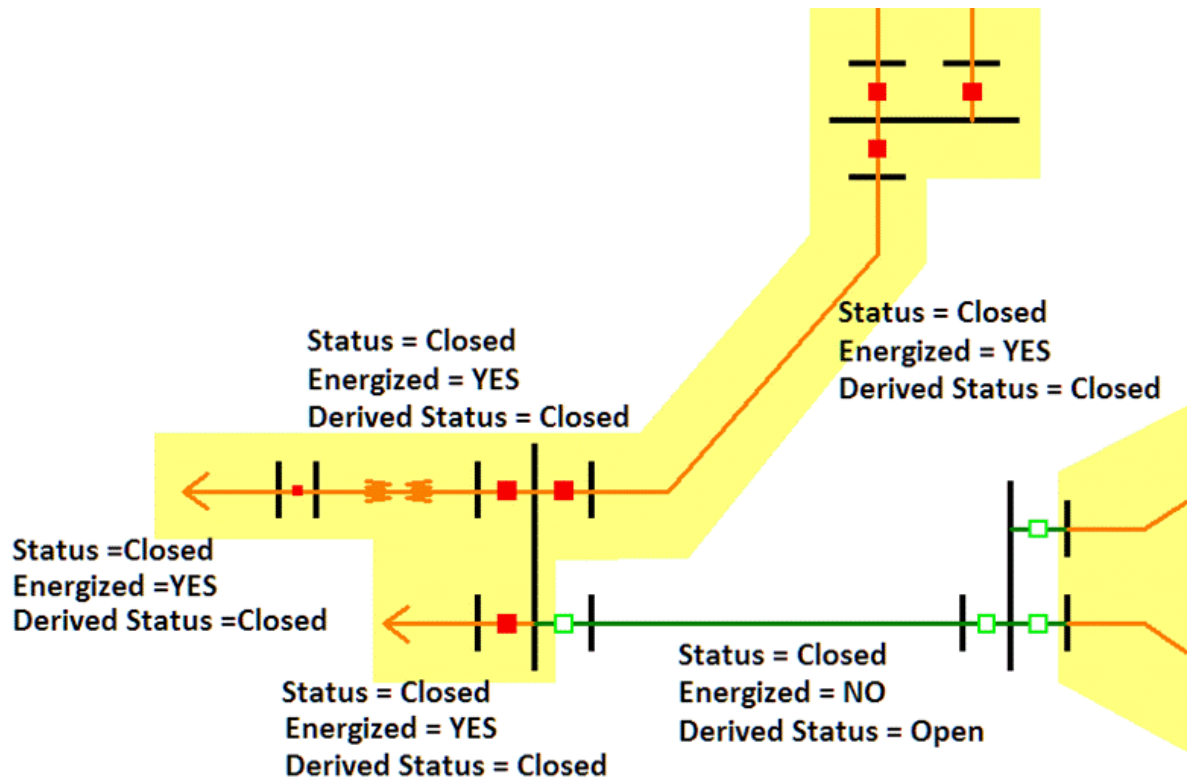
Derived Status is determined by the following:

- **Energized** = *YES* --> **Derived Status** = *Closed*
- **Status** = *Open* --> **Device Status** = *Open*
- **Energized** = *NO* and **Status** = *Closed* --> Ambiguous
 - Search starting at the device terminals across branches with **Status** = *Closed* looking for closed breakers
 - If closed breaker is found --> **Derived Status** = *Closed*
 - If no closed breakers are found --> **Derived Status** = *Open*

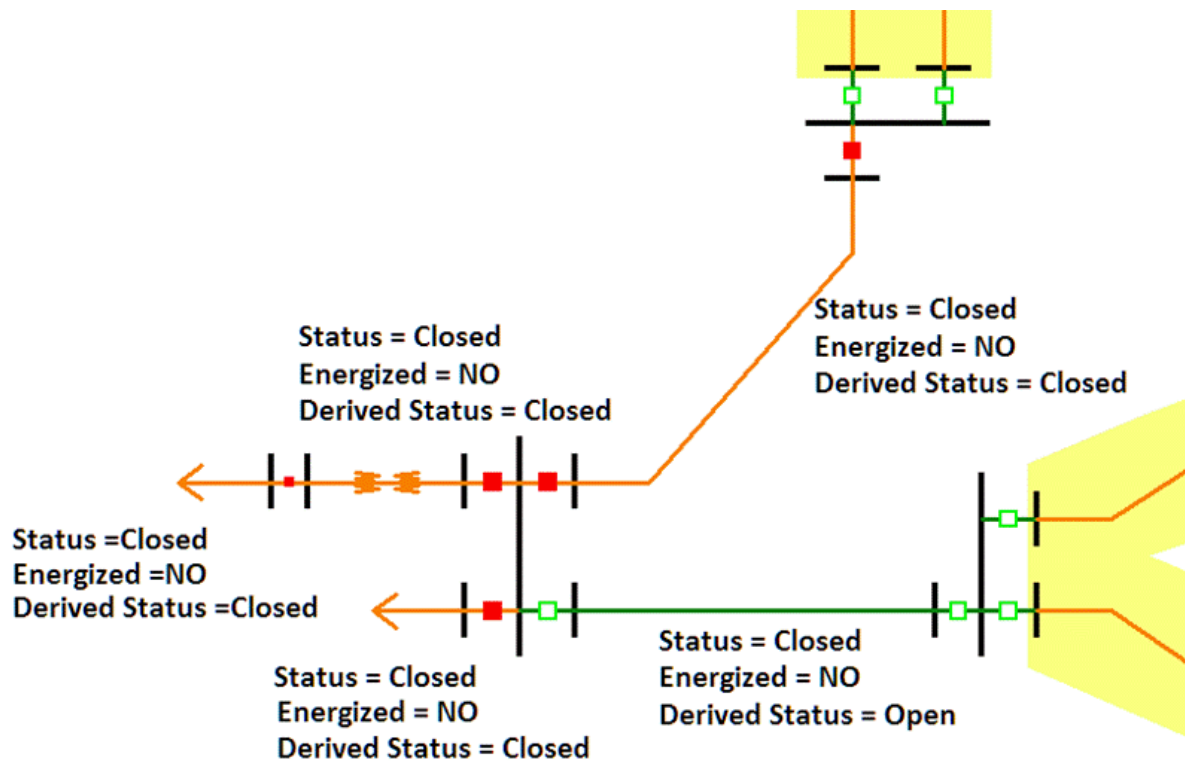
The following example illustrates how the **Derived Status** field is set for various devices based on their **Energized** and **Status** fields. In this example the yellow highlighted region represents the energize portion of the system. In the region that is shown, all devices are energized and all breakers are closed.



In the next example some breakers have been opened to open the lower line in the diagram. Again, the yellow highlighted region is the portion of the system that is energized. The **Status** of the line is still *Closed*, but it is no longer **Energized**. This is the ambiguous situation described above. The **Derived Status** is *Open* because searching out from the terminals of the line only open breakers are found before encountering any closed breakers.



In this last example, additional breakers have been opened, enlarging the region that is not energized. The upper line in the diagram has a **Status** of *Closed* and is not **Energized** due to the status of the open breakers. This again is the ambiguous situation described above. The **Derived Status** is *Closed* because closed breakers can be found by searching out from the terminals of the line before encountering any open breakers.



Transient Stability Overview

The Transient Stability tool is available as an add-on to the base Simulator package.
[Contact PowerWorld Corporation](#) for more details.

Transient stability studies analyze the system response to disturbances such as the loss of generation, line-switching operations, faults, and sudden load changes in the first several seconds following the disturbance. Following a disturbance, synchronous machine frequencies undergo transient deviations from synchronous frequency. The objective of a transient stability study is to determine whether or not machines will return to synchronous frequency following a disturbance.

For more detail on the transient stability modeling see the following links

- [Generator Modeling](#)
- [Wind Machine Modeling](#)
- [DC Transmission Line Modeling](#)
- [Load Characteristic and Relay Modeling](#)
- [Transient Stability Numerical Integration](#)
- [PlayIn Modeling](#)

For more detail on how to use the Transient Stability tool see the following links

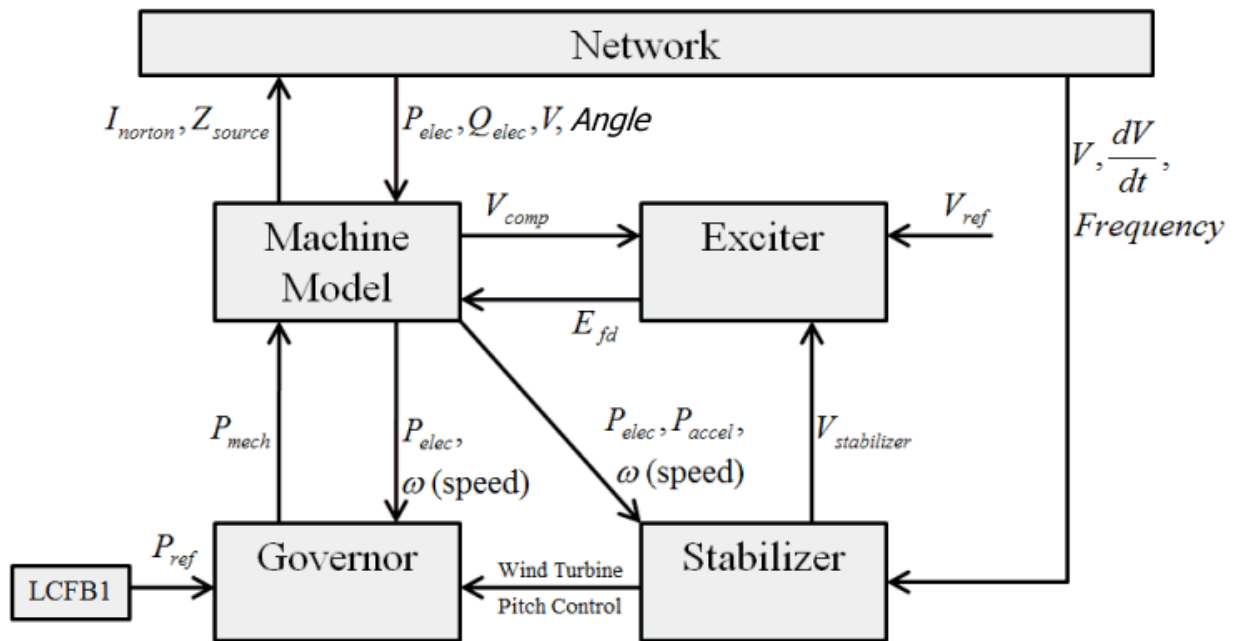
- [Transient Stability: Model Data Management](#)
 - [Stability tab of the Generator, Load, DC Line, etc... Dialogs](#)
 - [Transient Stability Case Information Drop-Down](#)
 - [Transient Stability Folder in the Model Explorer](#)
 - [Transient Stability Block Diagrams](#)
 - [Reading/Writing Transient Stability Data to various File Formats](#)
- [Transient Stability Analysis dialog](#)
 - [Simulation](#)
 - [Options](#)
 - [Results Storage](#)
 - [Plots](#)
 - [Results](#)
 - [Transient Limit Monitors](#)
 - [States/Manual Control](#)
 - [Validation](#)
 - [SMIB Eigenvalues](#)

Transient Stability Overview: Generator Models

Generator models are the most complex models in a transient stability tool. Generators can have many classes of dynamic model assigned to them including machine models, exciters, governors, stabilizers, and other models. You may define multiple models of each class of model to a generator, but for machine models, exciters, governors, and stabilizers, there can be only one *active* model. The relationship between these four models is shown in the figure below, along with the variables which are passed between the models. This is not an exhaustive list, but represents the typical relationships between the models.

Other stability models such as excitation limiters, Voltage Compensation, the turbine load controller (LCFB1) or a generator relay model may also be specified.

For wind generator models, similar relationships exist between a wind machine model, electrical model, mechanical model, and a pseudo-governor model. For more information about wind turbine modeling in Simulator, see the [Wind Turbine Modeling](#) help topic.



P_{elec} = Electrical Power

Q_{elec} = Electrical Reactive Power

V = Voltage at Terminal Bus

$\frac{dV}{dt}$ = Derivate of Voltage

V_{comp} = Compensated Voltage

P_{mech} = Mechanical Power

ω (speed) = Rotor Speed (often it's deviation from nominal speed)

P_{accel} = Accelerating Power

$V_{stabilizer}$ = Output of Stabilizer

V_{ref} = Exciter Control Setpoint (determined during initialization)

P_{ref} = Governor Control Setpoint (determined during initialization)

Governor Response Limits

There is a special generator field in Simulator called *Transient Stability|Governor Response Limits*. The field determines the response of the governor control limits during a transient stability run. Options are *Normal*, *Down Only*, or *Fixed*. When loading from an EPC file a Baseload flag value of 0 maps Normal, 1 maps to Down Only, and 2 or more maps to Fixed.

A value of *Normal* means that the limits specified in the governor model will be used for the simulation. A value of *Down Only* means that the upper limit is set equal to the initial condition value (and thus control can only go down). A value of *Fixed* means that both the upper and lower limits are set equal to the initial condition (and thus control will be approximately constant). Note that the power output of the generator can still vary for those turbines whose MW output is sensitive to speed (because the speed can obviously still vary).

Also, this field will always be shown as a default field when looking at a case information display showing a list of governors.

Special Accommodation for handling Line Drop Compensation

In modeling the WECC system, a stability issue was encountered at several hydro units such as John Day. The issue, which is described in the papers listed below, arises because of how the exciters for some hydro units with dual generators have been configured to enhance system voltage stability. Rather than regulating their terminal voltage, or a point internal to the generator (using compensation), the generator exciters have been configured using line drop compensation to regulate a point midway through the step-up transformer. This is illustrated in the below figure, which reproduces Figure 3 from the Murdoch/Sanchez-Gasca paper below. The advantage of this approach is it allows the generator exciters to directly regulate a point close to the transmission system (perhaps 50 to 80% of the way through the step-up transformer). If there is just one generator feeding through the transformer then this control can be modeled in the normal fashion. However when there are two generators, an extremely common configuration at hydro plants, physically the exciter inputs at each plant need to receive the currents from both machines in order to insure they are both regulating the same line drop compensated voltage. If this is not done, then the controls will end up fighting each other, driving the excitation current for one generator to its maximum and for the other generator to its minimum. Therefore to correctly model this situation, the PowerWorld Simulator transient stability excitation modeling code has been enhanced to include both generator currents in situations in which line drop compensation causes the regulated voltage to be beyond the generator's terminal. Otherwise the traditional code is used.

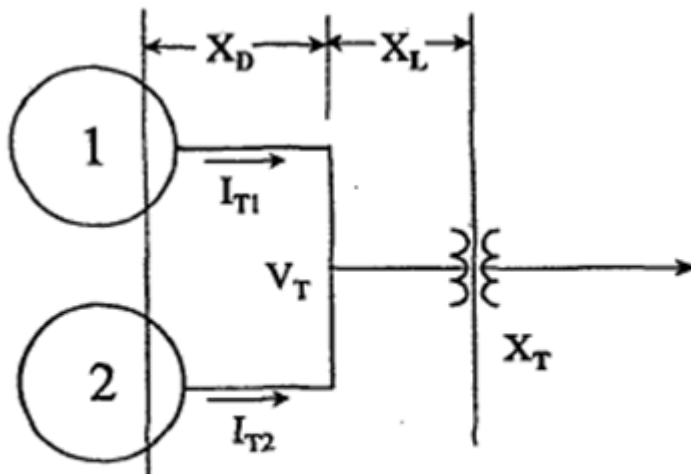


Fig. 3 - Two-Units Bussed Together, Showing Droop and LDC Compensation

NOTE:

When modeling a GENCLS model with $H=0$ and $D=0$ it is treated as an infinite bus.

A. Murdoch, J.J. Sanchez-Gasca, "Excitation Control for High Side Voltage Regulation," Proc. IEEE PES 2000 Summer Meeting, July 2000, pp. 285-289.

C.W. Taylor, "Line drop compensation, high side voltage control, secondary voltage control – why not control a generator like a static var compensator," Proc. IEEE PES 2000 Summer Meeting, July 2000, pp. 307-310

Transient Stability Overview: Wind Modeling

While strictly speaking, wind turbine models do not have an "exciter", "governor", or "stabilizer" in the way that a traditional synchronous generator model does, the relationships between the wind machine model, electrical model, mechanical model, and a pseudo-governor model are very similar. As a result, when looking at generators in the Simulator user interface, the various wind models will be listed under the traditional categories for synchronous machines of "machine model", "exciter", "governor", or "stabilizer". Examples are shown below.

Wind Machine Models	GENWRI, GENWTG, WT1G, WT1G1, WT2G1, WT3G, WT3G1, WT3GW, WT4G1
Wind Electrical Models listed as Exciters	EXWTG1, EXWTGE, WT2E1, WT3E, WT3E1, WT4E1
Wind Mechanical Models listed as Governors	WNDTGE, WNDTRB, WT12T1, WT1T, WT3T, WT3T1
Wind Pitch Control or Aerodynamic Models listed as Stabilizers	WT12A1, WT1P, WT3P, WT3P1

It will obviously not make any sense to configure a generator to use a wind machine model with a traditional synchronous machine exciter model and governor, so care should be taken not to setup such configurations. Presently Simulator does not check for such gross input errors.

A wind farm usually consists of many small (several MW) turbines. Each individual turbine's voltage is usually less than 1 kV (600 V is common) with a step-up transformer to increase the voltage to several dozen kV (34.5 kV common). The wind farm is usually modeled in aggregate requiring the aggregate model to account for impedance of collector system and then model per unit values at N times individual values. How accurate the aggregation is remains an open question.

There are four major types of wind turbine models used in transient stability studies which will each be considered in detail shortly.

- Type 1 : Induction generators with fixed rotor resistance
- Type 2 : Induction generators with variable rotor resistance
- Type 3 : Doubly-fed induction generators
- Type 4 : Full converter generators

Type 1: Induction generators with fixed rotor resistance

The most basic representation of a Type 1 wind turbine is as a conventional induction machine, however inertia is modeled with the machine. More detailed representations also include using a two-mass model (one mass for the generator and one for the turbine), and a pseudo governor. Inertia is sometimes modeled with the machine and sometimes as a governor in Simulator. A pseudo-governor is modeled as either a governor or a stabilizer in Simulator (depending on how inertia is modeled).

The MOTOR1 and GENIND models from the GE DYD file are the same, except for a sign convention on the current. They have integrated inertia. They cannot be used with other Type 1 governor/stabilizer models so no turbine dynamics are included. The CIMTR1, CIMTR2, CIMTR3, and CIMTR4 models from the PTI DYT file have the same restrictions as with the MOTOR1 and GENIND models .

The WT1G model is the GE DYD representation for a Type 1 wind turbine, while the WT1G1 is the PTI DYR model. Electrically they are quite similar to the GENIND model, except they do not include any inertia. Therefore they must be modeled with a WT1T (GE) or WT12T1 (PTI) model, both of which are included in the list of governors. The WT1T/WT12T1 models can represent the generator/turbine using either a one mass or two mass model. Both of these governors can also be used with Type 2 Wind Turbines.

The WT1P (GE) and WT12A1 (PTI) models represent a pseudo governor response model. These models are listed inside Simulator as a stabilizer models. The inputs to these models are machine speed and electrical output, while the output of the model is mechanical power. Again, these models can be used for both Type 1 and Type 2 wind turbines. It is recommended that you use the [machine / governor / stabilizer] grouping of [WT1G / WT1T / WT1P] or [WT1G1 / WT12T1 / WT12A1] to represent Type 1 wind machines.

Type 2 : Induction generators with variable rotor resistance

The Type 2 models augment the Type 1 by allowing for variable rotor resistance control in the wound rotor induction generator. This model is used to represent wind turbines such as the Vestas V80. Resistance control is represented by a PowerWorld Simulator using a model from the "exciter" list. Inertia is included with some machine models, or is included with a pseudo governor model. PowerWorld supports two classes of Type 2 models. From the GE DYD file we support the combination of the GENWRI machine, EXWTG1 exciter, and WNDTRB governor. From the PTI DYR file we support the combination of the WT2G1 machine, WT2E1 exciter, WT12T1 governor, and WT12A1 stabilizer.

Modeling Using GENWRI, EXWTG1, and WNDTRB models

The GENWRI models represents a single cage induction generator, and also includes a single mass inertia model with the machine. The initial operating slip must be given. From this, PowerWorld then calculates the necessary resistance to match this slip. The EXWTG1 exciter specifies the minimum and maximum external rotor resistance. The WNDTRB models the blade pitch control with an input of rotor speed and an output of Pmech. The resistance is set initially to match default slip of -0.04. During the fault the resistance increases to compensate for the increased speed.

Modeling Using WT2G1, WT2E1, WT12T1, and WT12A1 models

The W2G1 model is the PTI representation for a Type 2 wind turbine which should be similar to the GE models. These are all based on new standards which are being developed for wind turbine modeling. Electrically it is quite similar to the GENWRI model, except they do not include any inertia. Therefore they must be modeled with a WT1T (GE) or WT12T1 (PTI) model, both of which are included in Simulator as governor models. They also can use the WT1P or WT12A1 pseudo governors (which are included in Simulator as stabilizer models). The initial operating point is given by the R_Rot_Max field, which specifies the total rotor resistance. The external rotor resistance is controlled using the WT2E1 model, which is modeled in Simulator as an exciter.

The WT2E1 model controls the external resistance between values given by Rotrv_min and Rotrv_max. The inputs to WT2E1 are rotor speed and electrical power. Rotor speed is converted to an equivalent power using a piecewise linear slip-power curve that is entered with the WT2G1 model. The curve is assumed to have odd symmetry ($f(-x) = -f(x)$). An offset is added to model to get it to initialize to zero.

Type 3 : Doubly-fed induction generators

Modeling using the WT3G ,WT3E, WT3T and WT3P models

The GE DYD file supports a combination of models of a WT3G machine model, WT3E exciter, WT3T governor, and WT3P stabilizer to model a Type 3 wind generator. The WT3G model represents the generator behavior of the Type 3 and is the interface with network equations. The

WT3E model represents the reactive power control. The inputs to WT3E are the generator real and reactive power, and the voltages at the terminal and regulated bus, while the outputs are Eqcmd and Ipcmd. The WT3T model represents the mechanical equations. The WT3T inputs are the blade pitch and electrical power, while output is the rotor/turbine speeds. The WT3P model represents the pitch control. The WT3P input is the generator speed/power, while the output is pitch angle.

Modeling using the WT3G1, W3G2 ,WT3E1, WT3T1 and WT3P1 models

The PTI Type 3 models are very similar to the GE models except some default values are different. There are two machine models: WT3G1 and WT3G2. The WT3G2, an enhancement to the WT3G1, is recommended for new studies. WT3E1 is almost identical to WT3E (parameters in different order in the file format and PTI allows different negative rate limit). The WT3T1 is identical to WT3T. The WT3P1 is identical to WT3P (parameters in different order in the file format).

Detailed Models for common GE 1.5, 1.6, and 3.5 MW Turbines

GE provides more detailed models for their popular 1.5 MW, 1.6 MW and 3.5 MW turbines. These are modeled using the GEWTG (machine), EXWTGE (exciter), and WNDTGE (governor). The WNDTGE model combines the wind turbine model with the pitch control model; both one and two mass models are supported. PowerWorld provides the GE defaults for all three units for both single and double mass, and for either 60 or 50 Hz. When reading an EXWTGE model from a DYD file, PowerWorld automatically detects older record formats, reading parameters in the appropriate order for the format. Note that the latest updates on the GE wind/solar models are available at http://www.gepower.com/prod_serv/products/utility_software/en/ge_pslf/index.htm

Type 4 : Full Converter Models

To model full converter wind turbines, use the WT4G1 machine model and WT4E1 exciter model.

Transient Stability Overview: Loads

Most Load Characteristic Models as well as Load Relay Models in Simulator are allowed to apply to either a load, bus, owner, zone, area, or the entire system. During the dynamic simulation, a particular load record will use a the following priority when determining what to use as the load characteristic or load relay model in the stability simulation.

- If a load-specific model exist, this will be used
- Else if a bus-specific model exists at the terminal bus, this will be used
- Else if an owner-specific model exists for the load's owner, this will be used
- Else if an zone-specific model exists for the load's zone, this will be used
- Else if an area-specific model exists for the area's zone, this will be used
- Else if an system-specific model exists for the power system, this will be used
- Else the *Load Modeling* option specified on the [Transient Stability Dialog's Option\Power System Model](#) section will be used. (For relays if the default will be to not use a relay model at all)

Also note that for some types of models such as the MOTORW induction machine model, the model only applies to a specified percentage of the load. The remaining percentage of the load will use the algebraic model using the logic above.

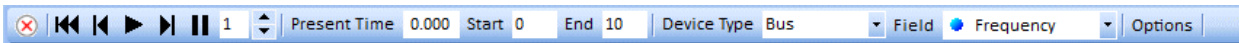
Transient Stability Overview : DC Transmission Lines

For transient stability simulations, DC transmission lines without a dynamic model specified are modeled as part of the algebraic network equations in much the same way they are modeled in a power flow solution for steady state analysis. The only exception when moving to a stability simulation is that all DC lines (as well as DC converters for multi-terminal DC lines) are assumed to operate in current control mode during the stability simulation. For those which are configured for power control mode in the steady state power flow input, Simulator will change the control mode to current and change the setpoint to the steady state current on the device.

Transient Contour Toolbar

The Transient Contour Toolbar is a special toolbar for generating a time-sequenced series of contour images based on the transient stability results. The results of the transient stability must have already been calculated and either [stored to RAM](#) or [stored to Hard Drive](#) as discussed in the [Results Storage](#). When generating a plot, an attempt will be made to retrieve from the results [stored to RAM](#). If results are not available for a particular plot series in RAM, then results will be retrieved from those values [stored to Hard Drive](#). If they are still not available then a plot will be generated which omits any plot series for which data can not be retrieved.

The Transient Contour Toolbar looks as follows.



This toolbar can be opened in one of two ways

1. Click the **Show Transient Contour Toolbar** option from the [Transient Stability Case Info Menu](#).
2. Click the **Show Transient Contour Toolbar** button at the bottom of the [Transient Stability Analysis Dialog](#).

The various buttons on this table are described from left to right below.

Close

Click this button to hide the Transient Contour Toolbar.

Reset

Click this button to reset the time of the data being contoured back to the start time stored with the results. A contour image is automatically generated at this start time then.

Back

Click this button to move the present time of the data being contoured backward the number of stored time points specified by the **Step per Animate** option.

Play

Click this to continue moving the contour visualization forward in time with each animation frame determined by the **Steps per Animate** option

Forward

Click this button to move the present time of the data being contoured forward by the number of stored time points specified by the **Step per Animate** option.

Pause

Click this button to

Steps per Animate

The entry of this integer value determines how time progresses when clicking the other buttons here.

Present Time

Shows the present time of the data being visualized by the contour

Start and End Time

Show the start and end time of the data available for visualization from the results.

Device Type and Field

Modifying the **Device Type** and **Field** in tandem determines which objects are to have values contoured and which field determines the contour value. The devices available will be an overlap between those devices which can be contoured and those devices for which results can be saved in the transient stability tool. This list of available fields will look very similar to the list available on the [Plot Designer](#).

Options

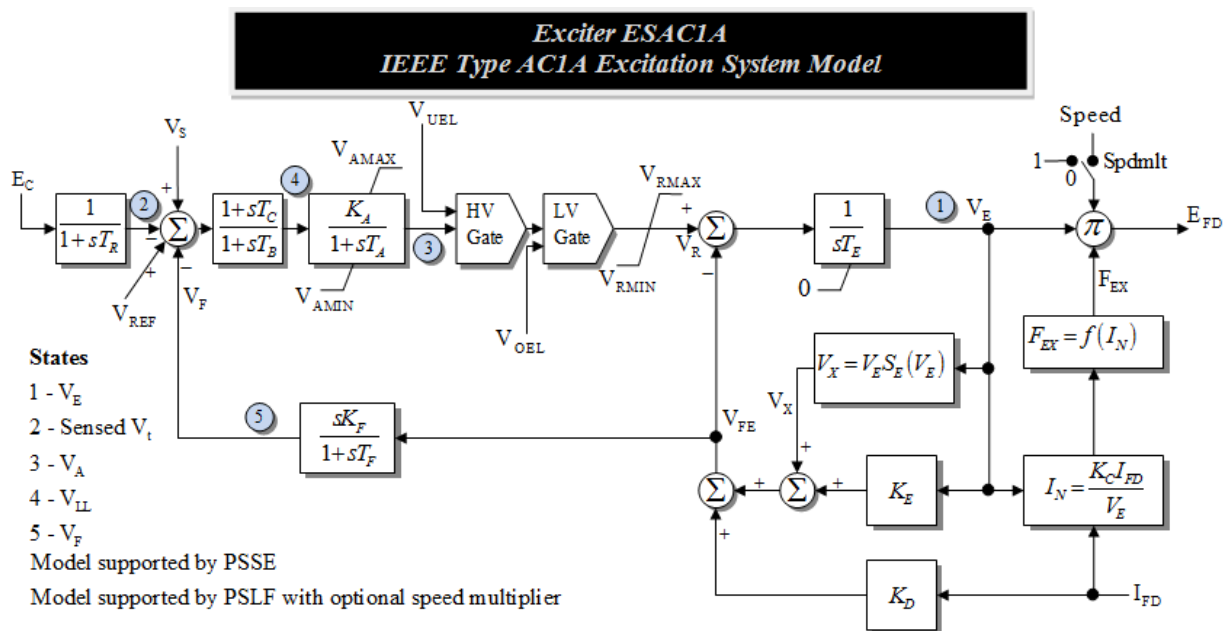
Clicking this button shows the [Contour Options Dialog](#) for a special Transient Contour Value field. This value will be populated according to the **Device Type** and **Field** options above.

Transient Stability Numerical Integration

The simulation of a transient stability event involves solving a set of differential and algebraic equations. PowerWorld Simulator uses a second order Runge-Kutta integration method to simulate the differential equations. A few special topics related to the integration method are as follows.

Handling Ignored States

In a commercial software package, each model must be implemented in a generic manner which accounts for all possible configurations that model may take. Because of this, Simulator must implement the maximum number of system state variables possible for each model. The software must then determine from the input data whether or not to ignore a particular state. As an example consider the ESAC1A exciter shown in the block diagram below.



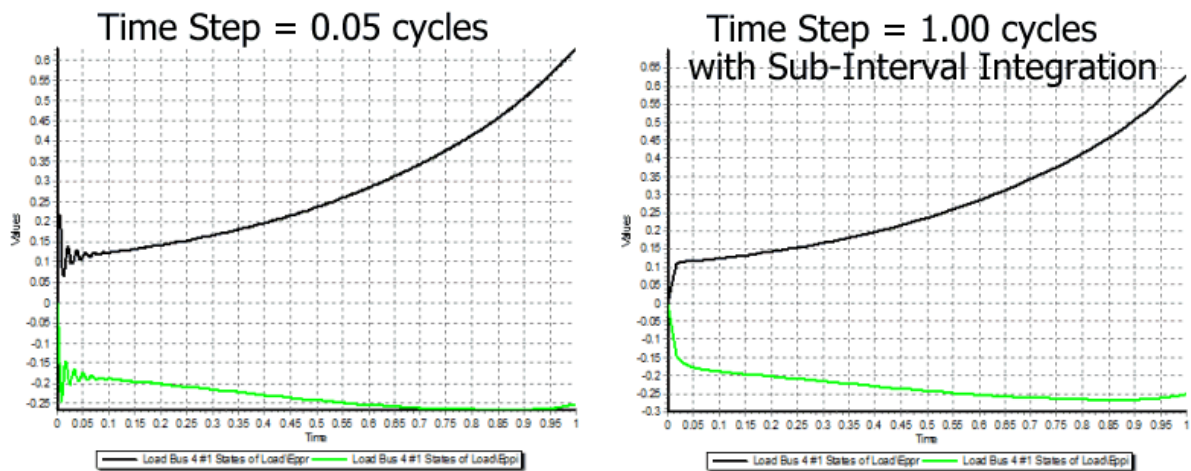
If the parameters T_c and T_b are both entered with a value of zero, this indicates that the state associated with the lead lag block will not be a dynamic state and instead will always be equal to the input value. Simulator's integration engine automatically handles these situations and the derivative of particular states will be reported as Ignored in the interface in these situations as shown in [Transient Stability Data: Object Dialogs](#). Also the propagation of values will automatically be handled. For example, any change to the input to the T_c block will immediately propagate to the output of that block.

Sub-Interval Integration

Like most commercial transient stability software, PowerWorld Simulator uses an explicit integration approach that alternates between solving the differential equations and solving the algebraic network power balance equations. This approach can be quite useful provided there are no model dynamics with time constants faster than the transient stability time step (using $\frac{1}{2}$ cycle). However for some transient stability models, such as the EXST1 exciter model, and during induction motor startup, these conditions can be violated. To avoid numerical instabilities with these models, we have implemented a technique known as sub-interval integration. When using this approach, within each transient stability time step the differential equations for some specific models are integrated using a much smaller time step with the assumption that during these shorter time steps the

algebraic (network) variables remain fixed. This approach has proven to be quite successful in integrating very fast dynamics.

As an example, the two figures below show the internal sub-transient voltages for an induction motor startup for one second of simulation time. In order to show the fast voltage transients that can occur during motor startup, the first figure is integrated with a very small time step (1/20 cycle) without the subinterval integration. Note the oscillations that occur during the first 0.1 seconds, with a frequency on the order of 60 Hz. The second figure repeats the first except using the new sub-interval functionality and a time step of one cycle. Because of the longer time step, the faster oscillations are no longer visible, but the response is numerically stable and is quite similar to the case with the much smaller time step. Without subinterval integration this case is numerically unstable with a time step of 1/5 cycle. Two key advantages of the subinterval integration approach with the longer time step are 1) that network algebraic equations are evaluated much less frequently, and 2) the subinterval integration is only applied to the handful of models for which it is required.



Transient Stability Overview: PlayIn

PlayIn models are used to play input data into the transient stability tool at a specified time. The models are set to be used as a generator, a reference signal or a governor. Respectively there are three types of PlayIn models available: PlayInGen, PlayInRef, PlayInGov and PlayInEx. The data for the models can be set manually using the **PlayIn Configuration** model type inside the Transient Stability folder on the [Model Explorer](#).

In the PlayIn Configuration model type you can insert a PlayIn model signal by its name and specified Offset to be applied when play in that particular model. To insert a new PlayIn model, right click on the PlayIn entry and choose Insert. This will open a dialog to enter the PlayIn model Name. Then after pressing OK the PlayIn Time offset can be specified. To insert data to a particular PlayIn Model right click on the entry and choose Show Dialog. This will open another dialog with a case information display to enter the data for the PlayIn Model. The Dialog have the PayInName which can be rename and also the Time Offset can be changed. Below there are two tabs: Signals Info and Signals.

In the Signal Info tab you specified the Signal Index as well as a respective name, scale, offset and filter. These signals are the signals that the PlayIn Model will have. For example, a PlayIn generator will need a signal for the amplitude of the Thevenin source and the frequency of the generator. The scale is used to scale the signal to a particular value. The offset will be applied to the signal to match the initial condition of the particular signal established in the initialization of the model. The filter values is a time constant, such as for a filter block $[1/(1+sT)]$, used to filter the signal.

In the Signal Tab the signal value can be specified for the particular signal index. To insert a new signal time right click in an entry and choose insert. This will open a dialog to specify a time for the signals.

The other easy way to enter PlayIn signals and new PlayIn models is to use an [auxiliary file](#) with the desired signals and the desired data values, offset, scale and filter values.

PlayInGen Model

The PlayIn Gen model is a generator model to play in data to the transient stability tool. The PlayIn Gen models a generator Thevenin voltage amplitude source in p.u. and the frequency amplitude in p.u.

The model can be entered in the generator dialog Machine Models [Stability Tab](#) and will required the following:

- PlayIn Model Signal applied for the particular generator.
- VIndex: Index of the Thevenin Voltage amplitude signal.
- FIndex: Index of the frequency amplitude signal.
- Rth: The Thevenin resistance of the generator in p.u.
- Xth: The Thevenin reactance of the generator in p.u.

PlayInRef Model

The PlayIn Gen model is a voltage regulator and governor reference model to play in data to the transient stability tool. The PlayInRef models a voltage regulator reference voltage in p.u. and the governor reference Pref (Speed-load reference) in p.u.

The model can be entered in the generator dialog Other Models [Stability Tab](#) and will required the following:

- PlayIn Model Signal applied for the particular generator.
- VIndex: Index of the voltage regulator reference signal.

- FIndex: Index of the governor reference signal.

PlayInGov Model

The PlayIn Gov model is a governor model to play in data to the transient stability tool. The PlayInGov models a governor mechanical power (Pmech) in p.u.

The model can be entered in the generator dialog Governors [Stability Tab](#) and will required the following:

- PlayIn Model Signal applied for the particular generator.
- FIndex: Index of the governor Pmech reference signal.

PlayInEx Model

The PlayIn Gov model is a n exciter model to play in data to the transient stability tool. The PlayInEx models an exciter field voltage in p.u.

The model can be entered in the generator dialog Exciters [Stability Tab](#) and will required the following:

- PlayIn Model Signal applied for the particular generator.
- FIndex: Index of the exciter field voltage signal.

Transient Stability: Model Data Management

The Transient Stability tool introduces a very large amount of additional input data which must be entered to define the dynamic models of the system. Interaction with this data is completely integrated with the rest of PowerWorld Simulator and is done primarily through the following methods

- [Stability tab of the Generator, Load, DC Line, etc... Dialogs](#)
- [Transient Stability Case Information Drop-Down](#)
- [Transient Stability Folder in the Model Explorer](#)
- [Transient Stability Block Diagrams](#)
- [Reading/Writing Transient Stability Data to various File Formats](#)

Details on these methods are found in the individual topics.

Transient Stability Data: Object Dialogs

On the various option dialogs for devices such as Generators, Load, Switched Shunts, DC Lines, and Transmission Lines, there will be a tab labeled Stability. On this tab you will be able to define the appropriate dynamic models for the device. An example of the generator dialog's Stability tab is shown as below.

Power and Voltage Control Costs OPF Faults Owners, Area, etc. Custom **Stability**

Machine Models Exciters Governors Stabilizers Other Models Step-up Transformer Terminal and State

Insert Delete Gen MVA Base 100.0 Show Block Diagram

Type Active - ESAC1A ☒ Active (only one may be active) Set to Defaults

Parameters

PU values shown/entered using device base of 100.0 MVA

Tr	0.0000	Te	0.8000	SE(E1)	0.0300
Tb	0.0000	Kf	0.0300	E2	4.0000
Tc	0.0000	Tf	1.0000	SE(E2)	0.1000
Ka	200.0000	Kc	0	Rate feedback constant, sec	0.2200
Ta	0.2200	Kd	0.3800	Vrmin	-14.0000
VaMax	7.5000	Ke	1.0000	Spdmlt	0.0000
VaMin	-7.5000	E1	3.0000		

Rate feedback constant, sec
Note: bold values indicate non-default values.

Hover Hint

All of these tabs have a similar layout which includes the following attributes

Insert Button

Clicking this button to insert a new model. A dialog will appear prompting you to choose which specific type of dynamic model to add for the object.

Delete Button

Clicking this button to delete a model.

Type Drop-Down

Shows a list of the presently defined models for this class. A model which is active will say *Active* before giving the name of the model. Normally there will only be one model of each class, but Simulator does permit you to define multiple models. For some models only one can be active at a time.

Active Check-box

There will be a check-box to make a dynamic model *Active* or *Not Active*. When a model is not active then it will be included in the dynamic simulation. Also note that for some classes of models you can have only one active model at a time. Governors, Exciters, Machine Models, and Stabilizers for generators all behave this way, however if you're defining a relay model you may have more than one active.

Show Block Diagram Button

When clicking this button, the [BlockDiagrams.pdf Adobe Acrobat file](#) will automatically be opened which contains all the dynamic model block diagrams. This PDF file will automatically be navigated to show the block diagram of the presently model.

Parameter List

At the bottom of the tab is a panel with numerical edit boxes allowing you to define all the input parameter values for the model. Next to each edit box is the name of the parameter. In addition to this if you hover the mouse over the edit box a hint will appear with a more detailed description of the parameter. The edit box, the non-default parameters will be highlighted as bold.

Special Options only on the Generator Dialogs

Set to Default

On the generator dialog there are options to change the parameters to PowerWorld's default settings.

Option to enter values on either the Device MVA base or the System MVA base

Normally per unit values for stability data are entered on the MVA base of the device. If you would like to change to enter data on the System MVA base instead, change this option. (Note: this option only affects the dialog entries. Entries in the case information displays will always be on the device MVA base.)

Several Model Class Tabs

Generator dialogs will list sub-tabs for Machine Models, Exciters, Governors, Stabilizers, and Other Models. Each of these sub-tabs has the same layout.

Step-up Transformer Tab

Information about an assumed step-up transformer may also be included with the generator and can be entered here.

Terminal and State Tab

After the transient stability run has been initialized on the Transient Stability Dialog, you may go to the Terminal and State Tab to View information about the terminal values and dynamic states. Under this tab will be several more sub-tabs as follows

Bus/Setpoint Values: Shows voltage and current on the system MVA base at the terminal bus. Shows the MW, Mvar and MVA being injected into the system at the terminal bus as well. Also shows the Exciter Setpoint (Vref) and Governor Setpoint (Pref) values if appropriate.

Terminal Values: Shows the Terminal Voltage and Current on the device per unit base. Mechanical MW, Accelerating MW, and Terminal MW. Also shows information about the internal rotor angle, Frequency Deviation (ω), and acceleration. Also shows information about the Field Voltage and Current and the Direct and Quadrature axis terminal and internal voltages and currents

Several Model Class State Tabs: tabs for Machine Models, Exciters, Governors, and Stabilizers will be available. Each will contain a list of the present states for the model as well as the derivatives of each state. For the derivative values, the states are sometimes ignored (such as when a time constant is zero). If this is true for the model, the derivative will be listed as *Ignored*.

Special Options only on the Load Dialog

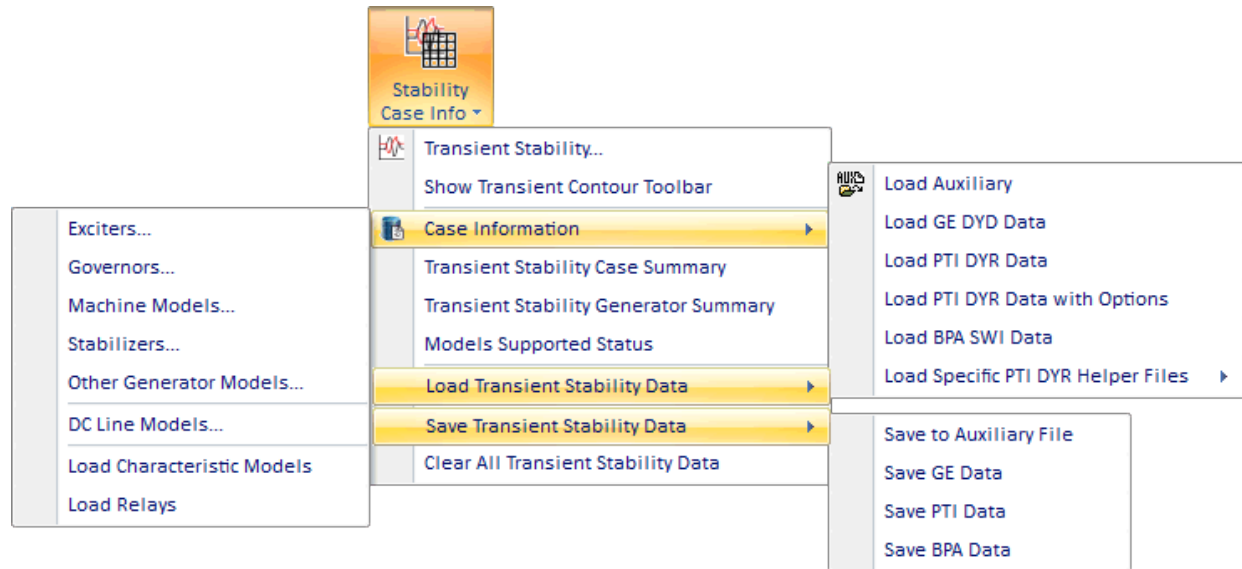
Type Drop-Down Modification

Because for a load model, the model can be inherited from the bus, owner, zone, area or system, the model listed under the Type drop-down may not be associated directly with the Load object.

If this is the case the model list will show the string *Inherited*. For inherited models you will not be able to edit the Parameter List directly. To edit the parameter list you will need to go to the dialog for that particular object in the [Transient Stability portion of the Model Explorer](#).

Transient Stability Case Info Menu

The Stability Case Info menu is available by selecting **Stability Case Info** in the **Transient Stability (TS)** ribbon group on the [Add Ons](#) ribbon tab.



The menu provides direct access to several case information displays that show default columns related to input and output of the transient stability tool. There are also options available for loading and saving transient stability models.

Transient Stability

Choose this option to open the [Transient Stability Analysis dialog](#).

Show Transient Contour Toolbar

Choose this option to make the [Transient Contour Toolbar](#) visible.

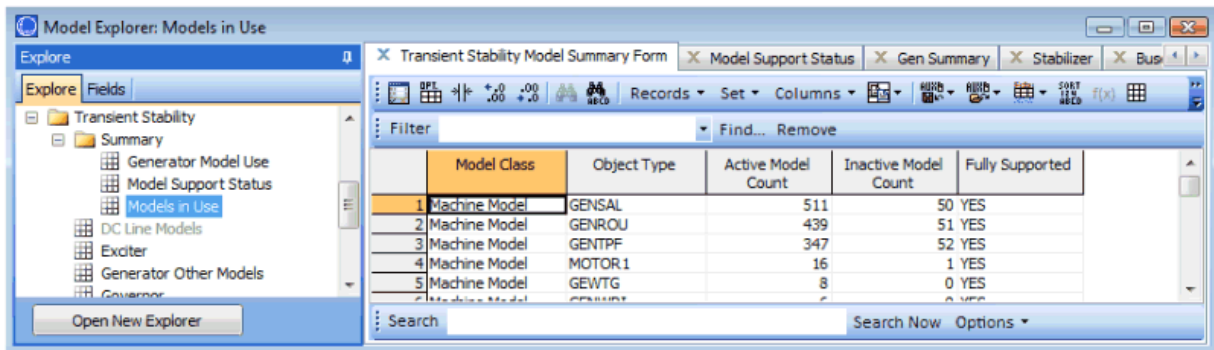
Case Information\Exciters, Governors, Machine Models, Stabilizers, DC Line Models, Load Characteristic Models, Load Relays

Choosing any of these options will open a case information display for the appropriate model type. This same information can be accessed through the **Transient Stability** folder on the [Model Explorer](#).

The stability case information displays can be used to change and define new models with the case. For a more complete description of the Model Explorer interaction with the transient stability models, see the [Transient Stability Model Explorer](#) help topic.

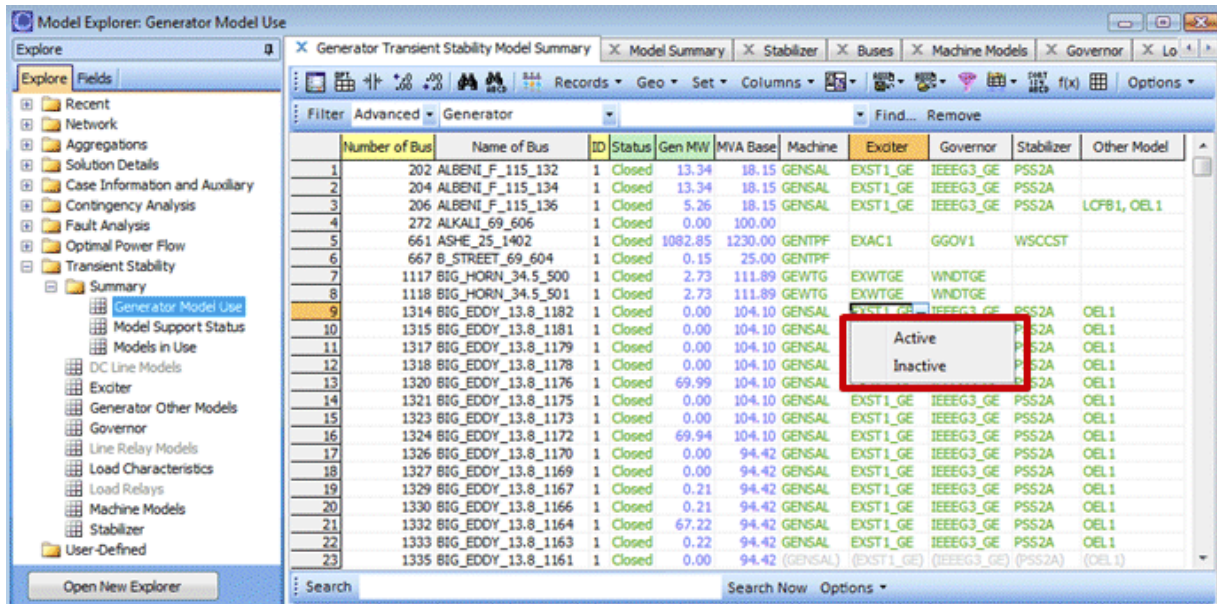
Transient Stability Case Summary

Choose this option to view a summary of transient stability models defined in the case. The summary includes the model name, whether the model is active or inactive, and whether or not the model is fully supported in Simulator. (For a small number of models, the model can be read by Simulator but is not actually supported for stability simulation. These models are marked as Fully Supported = NO).



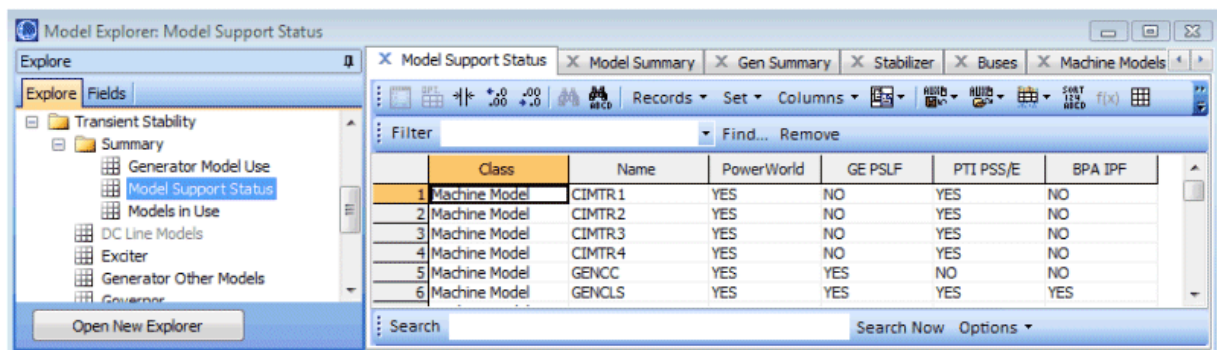
Transient Stability Generator Summary

Choose this option to view a summary of all generators in the case and any stability models that have been defined for each generator. Inactive models are shown in parenthesis. Use toggle to change active status of models. If multiple models exist for a particular model type, they are listed separated by commas.



Models Supported Status

Choose this option to view a list of all models which can be created in PowerWorld Simulator along with designations regarding what other software supports these models, and whether Simulator is able to include these models in the stability simulation.



Load/Save Transient Stability Data

Choose the appropriate option to save or load transient stability model data in the desired format. Models are supported in the Simulator auxiliary file format (*.aux), BPA dynamics data format (*.swi), GE dynamics data format (*.dyd or *.dyc), and PTI dynamics data format (*.dyr). MCRE *.rwm files, MTRLD *.dat files, GNET *.idv files, and BASEGEN *.dat files can also be applied. For more details about loading the GE DYD file see the [Reading DYD files](#) help topic.

Clear All Transient Stability Data

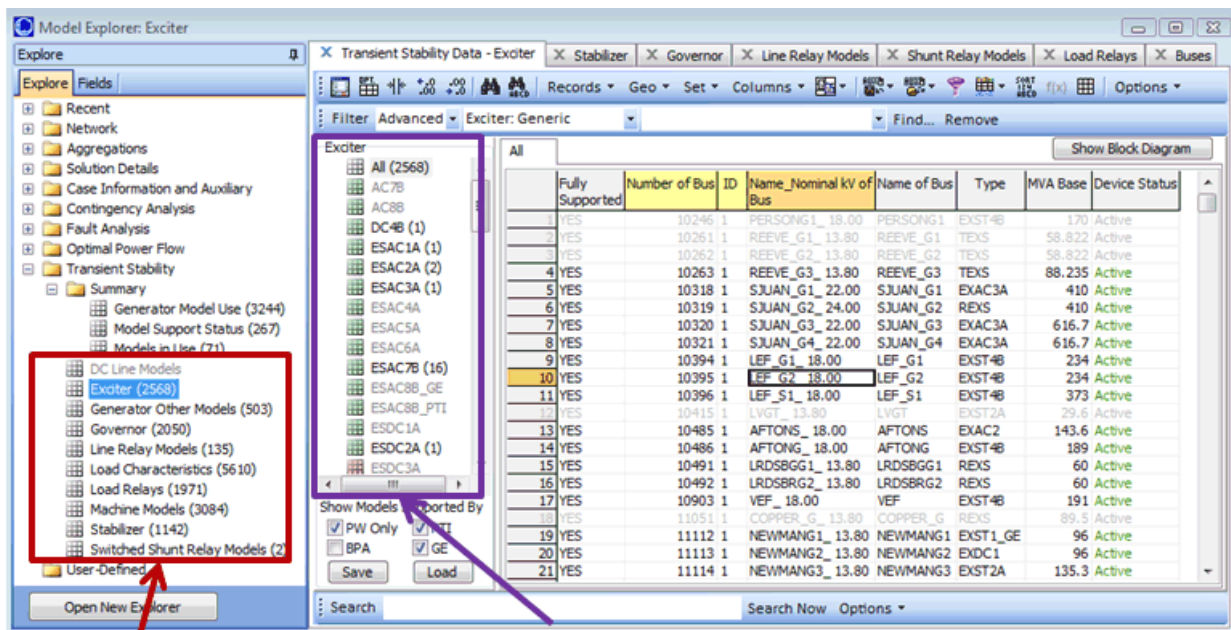
Choose this option to delete all transient stability models.

Transient Stability Model Explorer

On the Explore pane on the Model Explorer, there is a folder for Transient Stability. Immediately under this folder is a Summary folder containing links to Generator Model Use, Model Support Status, and Models in Use. These three features are discussed in more detail in the [Transient Stability Case Info Menu](#) topic. Under this folder is a list of the various classes of models such as Exciter, Stabilizer, Load Characteristic, etc. which are available in the transient stability tool. When choosing one of these entries, the right portion of the Model Explorer will be populated with a case information display showing models of that class. In addition there will be a new pane showing the specific types of that class of model. This pane has the following attributes

- Gray text indicates a model type which is not presently used by any device in the present power system case
- The number in parenthesis next to the model name indicates the number of models of that type which are used by a device in the present power system case
- Green icons indicate that the model is fully supported by the transient stability numerical integration Simulator.
- Red icons indicate that the model is not supported by the transient stability numerical integration in Simulator. When encountering a new model, PowerWorld will first configure the ability to define the model as well as read and write it from various file formats. After that the model will be implemented. Our intention over time is that all red icons will become green icons.
- At the bottom of the pane is a set of 4 check boxes showing **PW only**, **BPA**, **PTI**, and **GE**. These indicate which model types to show in the list depending on whether that model is supported by another software package. If you want to see only models which are supported by the BPA IPF program then uncheck the other three check boxes and check the BPA check box. For a complete listing of all models and where they are supported see the *Summary|Model Support Status* in the Model Explorer.
- The list of specific types will always include an entry at the top of **All** which lists all the models of that class. Below this will be a list of all the specific types of that model class. As you click on the various entries in the pane, the case information display will be populated accordingly.
- At the bottom are buttons to **Save** or **Load** [dynamic data from an external file](#).

The following shows such a listing for the generator Exciter models.



Class of Model Specific Types

All

On the listing which shows all models, there will only be columns which show identifiers for the object as well as two generic columns common to all models. The **Fully Supported** column is a YES/NO column which shows whether the model is supported by Simulator's transient stability numerical integration (the fast majority will say YES). The **Device Status** column will show either *Active* or *Not Active* depending on whether the device is specified to be used in the numerical integration.

Specific Model Type

When you choose a particular model type, additional columns will appear showing all the input parameters for that model type. This is depicted in the image below.

Machine Model

GENROU

Parameters

Show Block Diagram

Number of Bus	ID	Name_Nominal kv of Bus	Name of Bus	Type	MVA Base	Device Status	H	D	Ra	Xd	Xq	Xdp	Xqp	Xl	Tdop	Tqop	Tdopp	Tqopp	S(1,0)	S(1,2)	
1	10246	PERSONG1_18.00	PERSONG1	GENROU	170	Active	4.2	0	0	1.65	1.55	0.2	0.6	0.19	0.15	8.5	0.5	0.02	0.12	0.076	0.317
2	10261	REEVE_G1_13.80	REEVE_G1	GENROU	58.822	Active	4.22	0	0	1.73	1.6	0.285	0.8	0.2036	0.1	7.1	2	0.03	0.2	0.194	0.4997
3	10262	REEVE_G2_13.80	REEVE_G2	GENROU	58.822	Active	4.22	0	0	1.73	1.6	0.285	0.8	0.2036	0.1	7.1	2	0.03	0.2	0.194	0.4997
4	10263	REEVE_G3_13.80	REEVE_G3	GENROU	88.235	Active	3.48	0	0	1.76	1.52	0.24	0.7	0.2036	0.1	7	2	0.03	0.1	0.2196	0.5399
5	10394	LEF_G1_18.00	LEF_G1	GENROU	234	Active	4.87	0	0	2.25	1.825	0.275	0.85	0.2264	0.15	9	0.9	0.036	0.07	0.09	0.2687
6	10395	LEF_G2_18.00	LEF_G2	GENROU	234	Active	4.87	0	0	2.25	1.825	0.275	0.85	0.2264	0.15	9	0.9	0.036	0.07	0.09	0.2687
7	10396	LEF_S1_18.00	LEF_S1	GENROU	373	Active	2.91	0	0	2.27	1.7	0.33	0.85	0.27	0.231	7.5	0.9	0.036	0.07	0.065	0.5795
8	10415	LVGT_13.80	LVGT	GENROU	29.6	Active	7.4	0	0.0017	1.865	1.6	0.215	0.8	0.205	0.11	8	0.5	0.023	0.05	0.0992	0.348
9	10485	AFTONG_18.00	AFTONG	GENROU	143.6	Active	4.5	0	0.0038	1.744	1.6	0.256	0.4	0.17	0.1	7	0.75	0.03	0.05	0.05	0.3
10	10486	AFTONG_18.00	AFTONG	GENROU	189	Active	5.78	0	0.0025	1.715	1.615	0.22	0.4007	0.18	0.1236	8.5	0.539	0.055	0.083	0.0745	0.3077

Show Models Supported By

☒ PW Only

☒ FTI

☒ PRA

☒ FR

Show Block Diagram Button

At the top-right of the case information displays listing models will be the Show Block Diagram button. When clicking this button, the [BlockDiagrams.pdf Adobe Acrobat file](#) will automatically be opened which contains all the dynamic model block diagrams. This PDF file will automatically be navigated to show the block diagram of the presently selected model in the case information display.

Load Characteristic Models

Load models in Simulator are configured to apply to either a load, bus, owner, zone, area, or the entire case. This is discussed in more detail in [Transient Stability Overview](#). As a result you will see special folders at the bottom of the specific model type list for *Load-Specific*, *Bus-Specific*, *Owner-Specific*, *Zone-Specific*, *Area-Specific*, and *System-Specific*.

Transient Stability Data - Load Characteristic

Filter: Advanced Load Characteristic: Gene Find... Remove

Load Characteristic

- All (5725)
- CIM5
- CIM6
- CIMW
- CLOD
- CMPLD
- CMPLDW
- DLIGHT
- EXTL
- IEEL
- LDIFR
- MOTORW (5098)
- WSCC (627)
- Load-Specific (5098)
- Bus-Specific (604)
- Owner-Specific
- Zone-Specific (1)
- Area-Specific (21)
- System-Specific (1)

Show Models Supported By

☒ PW Only ☒ PTI

☐ BPA ☒ GE

Save Load

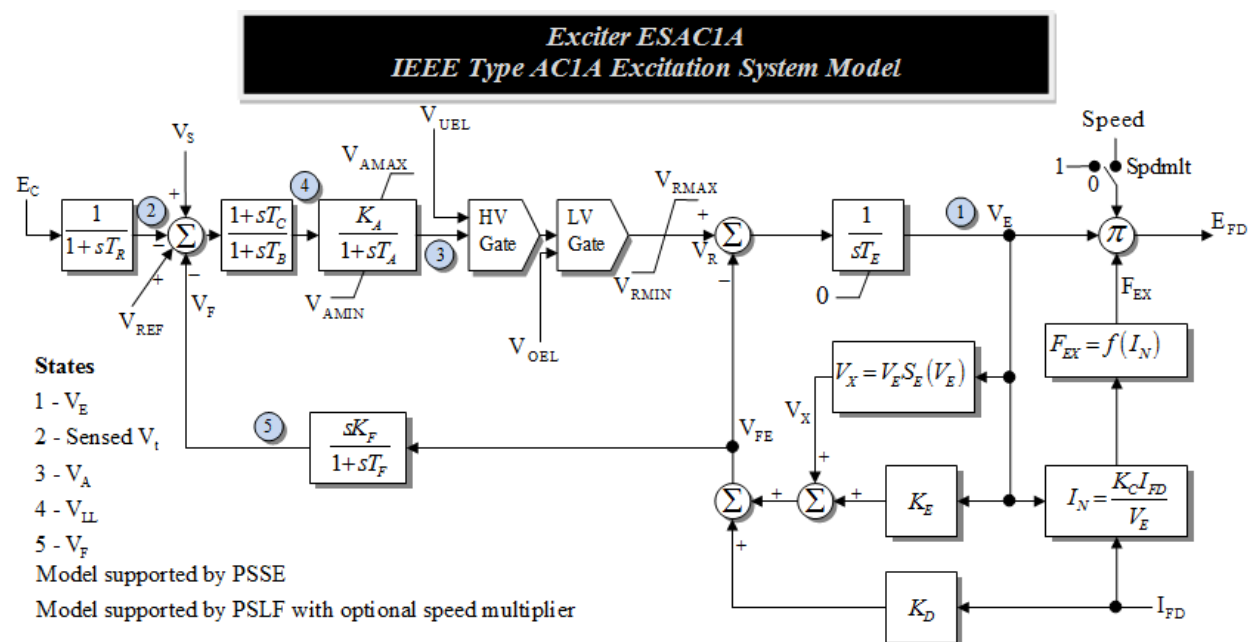
All

	Fully Supported	Element Type	Number	Name_Nominal kV	ID	Name	Type	Device Sta
3863	YES	Bus	50865	FHS 25_ 25.20		FHS 25	WSCC	Active
3864	YES	Bus	50866	AWT 60T2_ 60.00		AWT 60T2	WSCC	Active
3865	YES	Load	50868	WAH 25_ 25.20	1	WAH 25	MOTORW	Active
3866	YES	Bus	50868	WAH 25_ 25.20		WAH 25	WSCC	Active
3867	YES	Bus	50886	SFU 60T2_ 60.00		SFU 60T2	WSCC	Active
3868	YES	Bus	50891	SFU 60T1_ 60.00		SFU 60T1	WSCC	Active
3869	YES	Bus	50892	AWT 60T4_ 60.00		AWT 60T4	WSCC	Active
3870	YES	Load	50893	NLV 66_ 66.00	1	NLV 66	MOTORW	Active
3871	YES	Bus	50893	NLV 66_ 66.00		NLV 66	WSCC	Active
3872	YES	Bus	50906	NAK 12V1_ 12.60		NAK 12V1	WSCC	Active
3873	YES	Bus	50907	NDR 12V1_ 12.60		NDR 12V1	WSCC	Active
3874	YES	Load	50910	RVS 138_ 138.00	1	RVS 138	MOTORW	Active
3875	YES	Bus	50910	RVS 138_ 138.00		RVS 138	WSCC	Active
3876	YES	Load	50911	LBH 25T1_ 25.20	1	LBH 25T1	MOTORW	Active
3877	YES	Bus	50911	LBH 25T1_ 25.20		LBH 25T1	WSCC	Active
3878	YES	Load	50912	LBH 25T2_ 25.20	1	LBH 25T2	MOTORW	Active
3879	YES	Bus	50912	LBH 25T2_ 25.20		LBH 25T2	WSCC	Active
3880	YES	Load	50917	DKY 66_ 66.00	1	DKY 66	MOTORW	Active
3881	YES	Bus	50917	DKY 66_ 66.00		DKY 66	WSCC	Active
3882	YES	Bus	50919	IPR 25_ 25.20		IPR 25	WSCC	Active
3883	YES	Load	50927	CKY 25_ 25.20	1	CKY 25	MOTORW	Active
3884	YES	Bus	50927	CKY 25_ 25.20		CKY 25	WSCC	Active
3885	YES	Bus	50929	TOK 138_ 138.00		TOK 138	WSCC	Active

Search Search Now Options

Transient Stability Data: Block Diagrams

As part of the help documentation, Simulator includes an Adobe Acrobat File names *Block Diagrams.pdf* which contains all the block diagrams of dynamic models implemented in PowerWorld Simulator. This file can be automatically opened and the appropriate block diagram navigated to from both the [Transient Stability Case Information Displays](#) and from the [Stability Tab on the Generator, Load, DC Line, etc... dialogs](#). The format of these block diagrams is very standardized. The block diagram will appropriately label all the input parameters for the model type. There will also be a list of the dynamic states which exist for the model. These states will be labeled by number with a circle the number. In addition there will be a comment regarding whether another software product supports this model. The following is an example for the ESAC1A excitation system.



Transient Stability Data from External files

Transient Stability data can be read from several external file types including a PowerWorld Auxiliary File (AUX), PTI File (DYR), GE File (DYD), or BPA File (SWI). Access for loading these files is found in several places within Simulator.

- Menus within the [Application Menu](#)
- Bottom of the specific model pane on the [Transient Stability portion of the Model Explorer](#),
- Options within [Stability Case Info Menu](#),
- Buttons at the bottom of the [Transient Stability Dialog](#).

Save to Auxiliary

in the will prompt for a filename in which to save an [auxiliary file](#). Use this option to store any results and settings that need to be retained for future use or use with different power flow cases. Transient stability-specific results and option settings can optionally be saved with the PowerWorld binary file (*.pwb).

Load Auxiliary

Clicking this button will open a dialog to allow the user to select an [auxiliary file](#) to load. This is intended to be used for loading any relevant option settings to be used during the transient stability analysis. The dialog will be updated according to the option settings loaded from the file.

Save GE DYD Data, Save PTI DYR Data, Save BPA SWI Data

Choosing this option will allow you to save models to the appropriate file format. Note that only models which are supported by the specific format will be saved to the file. For a list of which models are supported by which format see the [Model Support Status case information display](#). When saving a DYD file, the model names are always stored lower case. When saving a DYR file, a GNET *.idv and a BASEGEN *.dat are also saved. The filename and path used are the same as for the DYR file, except "_GNET" and "_BASEGEN" are appended to the file names.

Load GE DYD Data, Load PTI DYR Data, Load BPA SWI Data

Choosing this option will allow you to load models from the GE DYD file format. Selecting the option "DYR file with options" will allow you to specify a MCRE *.rwm file to split up generators, a MTRLD *.dat file to split up motors, a GNET *.idv file to disable generator models, and a BASEGEN *.dat file to specify the Governor Response Limits flag for generators. "Load Specific PTI DYR Helper Files" allows you to apply those files on their own.

When loading from a GE DYD file a few special features have been implemented.

GE DYD: Special Handling for the Generator Baseload Flag

When loading GE EPC files, a flag exists in the EPC file with each generator record called the "Baseload flag". This field determines how governor limits are handled during a transient stability run. This feature is supported using Simulator generator field called *Transient Stability|Governor Response Limits*. For more information see the [Transient Stability Overview: Generator Models](#) topic.

GE DYD: Special Handling of EPCMOD models

In many example DYD files which PowerWorld has encountered, there are EPCMOD records which represents a user-defined model that represent a Series Capacitor Relay and a Capacitor Relay Model. Because these models were so common, PowerWorld added two new kinds of relays to our

model suite. The CAPRELAY model can be assigned to a switched shunt record. This represents a relay which can open and close a switched shunt based on a definite time relay measuring voltage. A SERIESCAPRELAY model can be assigned to a branch and used to bypass and place back inservice a series capacitor branch.

When parsing the DYD file, Simulator will detect an EPCMOD record and if that record refers to the EPCL program "MSC01.p", then we automatically assume it represents a switched shunt relay and create a CAPRELAY model. An appropriate log message will be written indicating this has occurred such as the following: "Info: An EPCMOD record using "MSC01.P" was found. Simulator will attempt to read following as a CAPRELAY: 30.000 1.2000".

When parsing the DYD file, Simulator will detect an EPCMOD record and if that record refers to the EPCL program "MSC02_R1.p", then we automatically assume it represents a series cap relay and create a SERIESCAPRELAY model. An appropriate log message will be written indicating this has occurred.

GE DYD: Special Handling of GENCC models

In the DYD often represent cross-compound generators in a special manner. A cross-compound generator represents two generators which both operate off the same steam plant. In a DYD file a cross-compound plant will be represented by 2 machine models, 2 exciters, 2 stabilizers, and 1 governor. In DYD files, the governor will be either the IEEE1G1 or CRMGV governors. Ideally, the power flow model represented by the EPC file will model these two generators explicitly and thus the DYD file will refer to the two generators when specifying the machines, exciters, stabilizer and governor. The DYD file however also supports a special machine model called the GENCC which is really just a GENTPF model but it signals to the tool reading the DYD file that two generators in the DYD file may represent one generator in the EPC file.

To accommodate this situation, Simulator's DYD parser has been written to first go through the entire DYD file and look for any bus which has only one generator in the existing case (EPC file), but the DYD file has 2 GENCC records at the bus with IDs that do NOT match any existing generator. If this situation is found, the user will be asked whether they would like to automatically split the existing generator into two generators to accommodate this modeling. If you choose yes, then the case will be permanently modified to split the generators into two. An appropriate log message will be written indicating this has occurred such as the following: "Info: New generator created at ALAMT3 G (24003) #L and existing generator ID changed to #H because 2 GENCC records found for Bus ALAMT3 G (24003) with IDs "H" and "L", but only one generator exists in the case at this bus with id "3"."

Also note that the GENCC records in the DYD file have parameters Pfac and Qfac which determine the percentage of MW and Mvar assigned to each generator. When the generators are split into 2 units, the following fields are multiplied by appropriate normalized Pfac and Qfac values: MW output, Max MW, Min MW, Mvar output, Max Mvar, and Min Mvar. The MVABase will also be multiplied but only if the GENCC record did not include this as part of its definition (normally it is included in the DYD file).

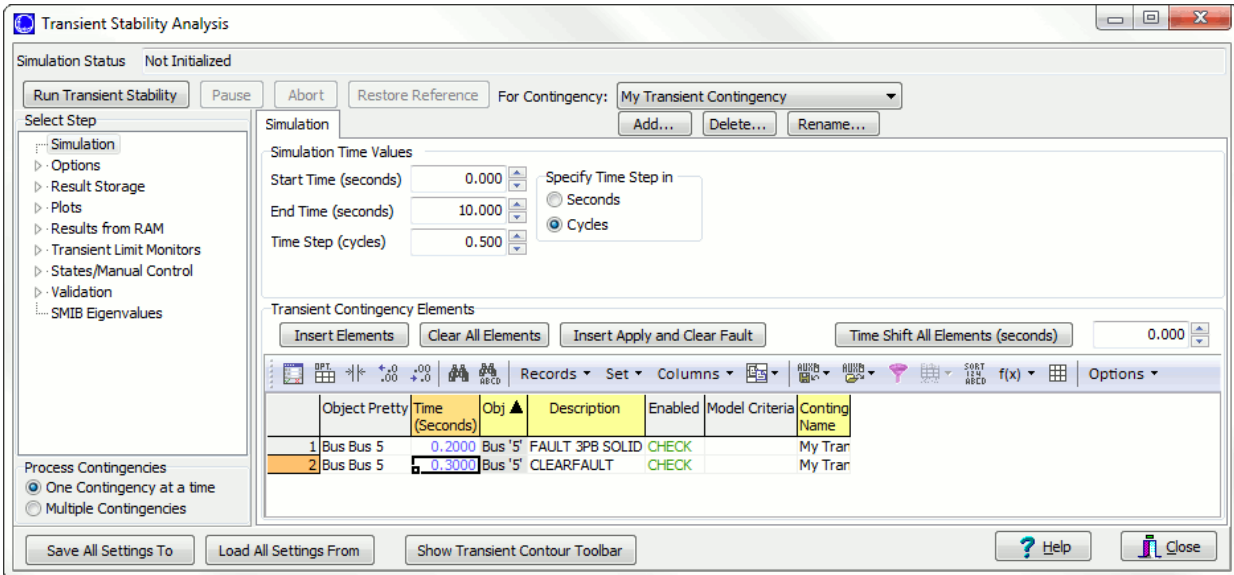
GE DYD: Appending changes

When loading files with the extension ".dyc", the contents are automatically appended to the existing models.

Transient Stability Analysis Dialog

To display this dialog, go to the [Add Ons](#) ribbon tab and select **Transient Stability** from the **Transient Stability (TS)** ribbon group.

All of the help documentation will discuss how this dialog functions when you are simulating only a single transient contingency event. When changing the **Process Contingencies** option to *Multiple Contingencies* instead, the dialog will change in small ways throughout. For a description of how the dialog behaves differently when running multiple contingencies, see the [Transient Stability Dialog Multiple Contingencies](#) help topic.



The dialog is broken down into several steps designed to take the user through the setup, processing, and analysis of results. The pages are as follows:

Simulation	Define the transient contingency elements which describe the event to simulate. Specify start time, end time, and time step.
Options	Specify the options: General , Power System Model , Result Options , and Generic Limit Monitors
Results Storage	Specify what results to store during the simulation for viewing , plotting , and contouring later. Also specify whether to store to RAM , store to Hard-Drive , both, or neither (when using Transient Limit Monitors appropriately, it may be unnecessary to store the actual numerical results.)
Plots	Build plot descriptions to both view results and specify what to store to RAM
Results	View the numeric results of the simulation as well as a summary of events that occurred

[Transient Limit Monitors](#)

Define transient limit monitors to look for violations of stability criteria during the simulation. For example, look for a specific drop in frequency that lasts for period of time.

[States/Manual Control](#)

Provides ability to manual step through the simulation a few time steps at a time. Also gives access to view the initialized stability state variables and initial state limit violations.

[Validation](#)

Provides options to validate the stability data specified for possible errors and warning.

[SMIB Eigenvalues](#)

Perform Single Machine Infinite Bus (SMIB) analysis on each generator in the system to help debug potential model errors which lead to instability.

There are several buttons at the top of the dialog that are normally available regardless of the selected page.



Run Transient Stability

Click this button to start the analysis once all options and models have been set.

Pause

Click this button to pause a study that is in progress.

Abort

Click this button to stop a study that is in progress.

Simulation Status

Indicates the progress of a study.

For Contingency:

This drop-down determines which contingency is presently being studied. See the [Transient Stability Dialog Multiple Contingencies](#) for processing multiple transient contingencies simultaneously.

Add, Delete, Rename

Click these buttons to Add, Delete, or Rename Transient Contingencies. See the [Simulation](#) topic for more details.

There are several buttons at the bottom of the dialog that are available regardless of the selected page.

Save All Settings To...

Clicking this button will show a drop-down menu for saving transient stability data to an external file. More details are found in the [Transient Stability Data from External files](#) help topic.

Load All Settings From...

Clicking this button will show a drop-down menu for loading transient stability data from an external file. More details are found in the [Transient Stability Data from External files](#) help topic.

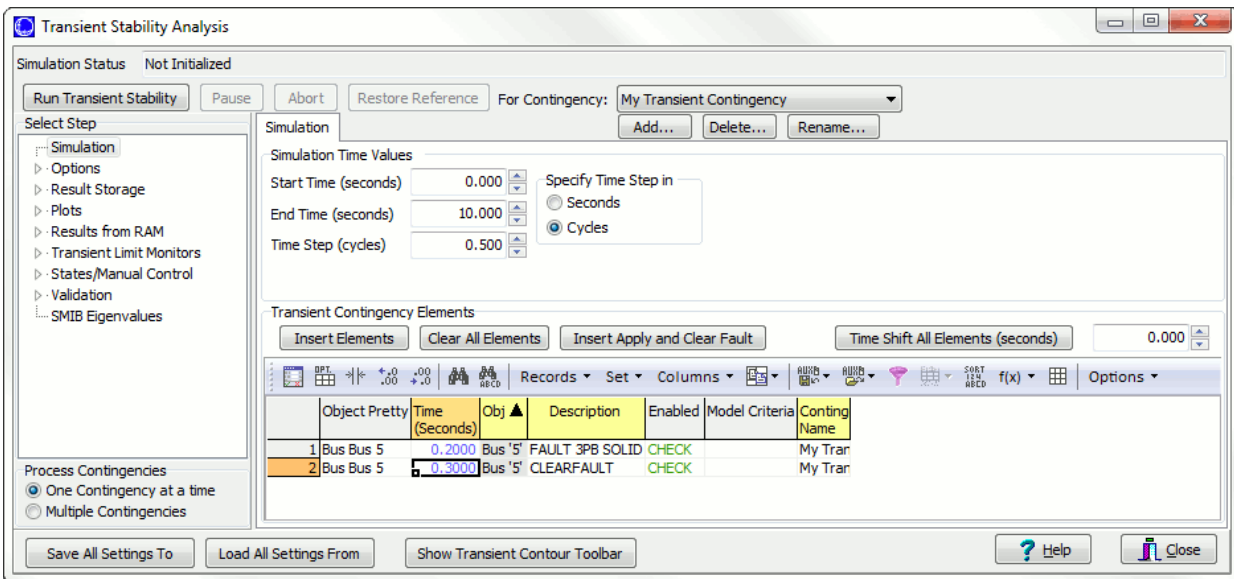
Show Transient Contour Toolbar

Clicking this button will automatically add a toolbar to the bottom of the main window in Simulator. This toolbar allows you to visualize the results of the stability run through a series of contour images. It's operating is described in more detail in the [Transient Contour Toolbar](#) Topic.

Transient Stability Analysis: Simulation

The Simulation Control page is available on the [Transient Stability Analysis dialog](#).

The Transient Stability Dialog allows you to define a particular transient stability event which you want to study. PowerWorld refers to this as a *Transient Stability Contingency* because in concept it is similar to the contingency analysis run using power flow solutions. The goal is to simulate a particular contingency to see if it causes any problems for the system. The contingency will consist of one or more events. For example, a transient contingency with only one event might be to model the outage of a generator with the goal of seeing how the governor response of the remaining generators handles this. Another contingency might be to model a fault at a bus which requires two events: one to initiate the fault and a second to clear the fault.



Defining a Transient Contingency

When the **Process Contingencies** option is set to *One Contingency at a Time*, the dialog will appear as depicted below. When defining multiple transient contingencies see [Transient Stability: Running Multiple Contingencies](#).

Use the Add, Delete, and Rename buttons to manage your transient contingencies. The drop-down allows you to choose the presently active transient contingency. Transient Contingencies differ from the power-flow based contingency analysis most importantly by introducing the concept of timing. Transient Contingencies require you to specify a Start Time and an End Time for the simulation, both specified in seconds. You must also specify a simulation time step which will be used by the numerical integration software. The following parameters for the transient contingencies are specified at the top of the dialog.

Add..

Click to add a new transient contingencies. The new contingency will automatically be given an unused name starting with "My Transient Contingency". Note that when choosing to [Store Results to Hard Drive](#), then name of the transient contingency will determine the name of the file to which results are stored.

Delete...

Click to Delete the present contingency. Note that at least one transient contingency must always exist. If you choose to delete the only transient contingency defined, then a new empty transient contingency names "My Transient Contingency" will automatically be created.

Rename

Click this button to be prompted to rename the present transient contingency.

Start Time, End Time

Specify the start and end time of the analysis in seconds.

Time Step

The time step is entered in either seconds or cycles (60 cycles/second) depending on the setting of the **Specify Time Step in** option. This value will change to reflect the appropriate units if the **Specify Time Step in** option is changed.

Specify Time Step in

The **Time Step** can either be specified in seconds or cycles. Changing this option will update the value specified in **Time Step** to reflect the appropriate units. The time step may be specified in seconds or in a number of cycles. For instance, for large studies PowerWorld recommends that you use a half cycle as a time step

Defining Transient Contingency Elements

Once you've specified this timing, you must then specify the transient contingency elements. Again they are similar to contingency elements for power-flow based simulation, except that these events must have a time associated with them. An example shown in the figure above is a transient contingency which faults a bus at 0.2 seconds and then clears the fault at 0.3 seconds.

Insert Elements

Click this button to open the [Transient Stability Contingency Element Dialog](#).

Clear All Elements

Click this button to delete all currently defined events.

Insert Apply and Clear Fault

Clicking this button also opens [Transient Stability Contingency Element Dialog](#), however the dialog will be modified to only allow the specification of a bus fault or a branch line fault. There will also be an option on the dialog to enter the Clearing Time in Seconds. This provide a convenient way to more quickly enter a fault and the clearing time of that fault instead of having to enter the dialog twice.

Element Table

This table provides a listing of all transient stability elements that are currently defined. This is a [case information display](#) and provides the case information toolbar and local right-click menu for options. New events can be inserted through the **Insert** menu/toolbar item. Events can be deleted through the **Delete** menu/toolbar item. Use the **Show Dialog** option to open the [Transient Stability Contingency Element Dialog](#) to modify the event. Model criteria can be applied to transient contingency actions in this table as well. The action will then only be applied if the initial conditions meet the model criteria.

Transient Stability Contingency Element Dialog

The Transient Stability Contingency Element Dialog is used to specify specific transient stability events and when they occur during the analysis.

Transient Stability Contingency Element Dialog

Description: 1.000: [Bus Bus 7] FAULT 3PB SOLID

Insert Delete

Object Type

- ☐ Simulation
- ☒ Bus
- ☐ Generator
- ☐ Load
- ☐ Switched Shunt
- ☐ AC Line/Transformer
- ☐ DC Line
- ☐ Injection Group

Choose the Element

Sort by: ☐ Name ☒ Number

Filter: Advanced Bus

☒ ☐ Use Area/Zone Filters Define/Find... Re

5 (Bus 5) [230 kV]
6 (Bus 6) [230 kV]
7 (Bus 7) [230 kV]
8 (Bus 8) [230 kV]
9 (Bus 9) [230 kV]

Time

Time (Seconds): 1.00000

Description

Type

- ☒ Apply Fault
- ☐ Clear Fault
- ☐ Open

Parameters

Fault Type: Balanced 3 Phase

Fault Across: Solid

PU Resistance: 0.000

PU Reactance: 0.000

OK Save ? Help X Cancel

Event Description

Drop-down containing a list of all currently defined events. An even description will be created automatically based on the event time, event object, and even action. This drop-down can be used to switch between events and make any necessary modifications.

Object Type

Specify the object type for the event. The type of object chosen will change the type of events that can occur.

Choose the Element

This list will be updated as the **Object Type** is changed. This list contains all of the available objects for the selected **Object Type**. Select a specific object from this list for which the event will be applied.

Event Time

Specify the time in seconds at which the event occurs.

Event Type, Event Parameters

The type of event and parameters will change depending on the Object Type chosen. Valid options are:

Bus

Apply Fault

Apply the specified fault type.

Fault Type - Balanced 3 Phase, Single Line to Ground, Line to Line, or Double Line to Ground

Fault Across - Solid, with Impedance and specifying **PU Resistance** and **PU Reactance**, or with Admittance specifying **PU Conductance** and **PU Susceptance**

Clear Fault

Clear a fault.

Open

Opens all AC lines connected to a bus.

Generator

Open

Open the generator.

Close

Close the generator.

Ramp Values

Ramp the output of the generator as defined by the **Action Field**. Options for the Action Field are MVA at Bus, Exciter Setpoint (Vref) and Governor Setpoint (Pref). Specify that the output be ramped **By Value** or **Percentage**. Specify the **End Time** in seconds at which the ramping stops.

Set Values

Set the output of the generator as defined by the **Action Field**. Options for the Action Field are MVA at Bus, Exciter Setpoint (Vref) and Governor Setpoint (Pref). Specify that the output be set **By Value** or **Percentage**.

Load

Open, Close

Specify that the load either be opened or closed.

Set Values

Set the output of the generator as defined by the **Action Field**. Option for the Action Field is MW, Exciter Setpoint (Vref) and Governor Setpoint (Pref). Specify that the output be set **By Value** or **Percentage**.

Switched Shunt

Open, Close

Specify that the load either be opened or closed.

AC Line/Transformer

Apply Fault

Apply the specified fault type.

Fault Type - Balanced 3 Phase, Single Line to Ground, Line to Line, or Double Line to Ground

Fault Across - Solid, with Impedance and specifying **PU Resistance** and **PU Reactance**, or with Admittance specifying **PU Conductance** and **PU Susceptance**

Percent Location (near to far) - Specify the location of the fault as a percentage of the distance from the near end of the line. A fault at the near end would be a percentage of 0% and a fault at the far end would be percentage of 100%.

Clear Fault

Clear a fault.

Open

Which End - Both Ends, From End Only, To End Only, One Phase Open

Close

Which End - Both Ends, From End Only, To End Only

Bypass, and Not Bypass

Use this option to either Bypass or Not Bypass a branch. Normally we would expect this branch to be a series capacitor or reactor.

DC Line

Open

Open the entire DC line.

Injection Group

Open

P or Q - MW, MVar

Device - Generation, Loads, Shunts

MW or MVar - Amount to open. The amount to open will be based on the initial condition of the device (not the transient condition), and devices will be opened in order of highest participation factor to lowest participation until at least the amount specified has been exceeded.

OK

Click this button to accept any changes and close the dialog.

Save

Click this button to save any modifications to the current event. The dialog will remain open.

Insert

Click this button to create a new event with the specified parameters.

Delete

Click this button to delete the current event as defined by the **Event Description**.

Help

Click this button to open this help topic.

Cancel

Close the dialog without saving any of the changes.

Transient Stability Analysis: Options

The Options page is found on the [Transient Stability Analysis dialog](#). The Options page is broken down into several sub-categories of options described as follows.

[General](#) : options for various user interface choices of how data is displays and how the user interacts

[Power System Model](#) : options for the actual transient stability numerical simulation

[Result Options](#) : options for how the results are stored during the simulation

[Generic Limit Monitors](#) : special options that apply generic limit monitors to all objects during the simulation

Transient Stability Analysis Options: General

The General sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#). This contains various choices of how data is displayed in the user interface and how you interact with the user interface.

The screenshot shows the 'Options' dialog box with the 'General' tab selected. A note at the top states: 'Note: Changes made to option entries are saved immediately and will be applied during the next transient stability run.' The 'General' tab is active, showing options for MVA Base, Automatic Update and Transfer Results, and Identify Buses. The 'MVA Base for Input/Display of Generator Values' section has two radio buttons: 'Use Individual Generator MVA Base' (selected) and 'Use Common Power Flow MVA Base'. The 'Automatic Update and Transfer Results to Power Flow Options' section includes a 'Display Update/Check Pause Interval (Timesteps)' spinner set to 10, and several checkboxes: 'Update Displays after Interval Check' (unchecked), 'Transfer Results to Power Flow after Interval Check' (unchecked), 'Transfer Results to Power Flow On Manual TimeStep' (checked), 'Transfer Results to Power Flow On Run Until Specified Time' (checked), 'Transfer Results Immediately for Events' (checked), 'Show Results Page When Done' (checked), 'Run at Rate Proportional to Real-Time; Slowdown =' (set to 5.0), and 'Allow Interactive Power Flow Control' (unchecked). The 'Identify Buses in Events and Results by' section has a list of radio buttons: 'Name' (selected), 'Number', 'Name (Number)', 'Number (Name)', 'Name and kV', 'Number and kV', 'Name (Number) and kV', and 'Number (Name) and kV'.

MVA Base for Input/Display of Generator Values

Specify the MVA base to use for generators: either the MVA base value entered for each individual generator or the MVA base value specified for the case.

Identify Buses in Events and Results by

Choose how to identify any buses that appear in transient stability events and results. Buses can be identified by number or name or a combination of both. The nominal kV of the bus can also be used as part of the identifier.

When Case has a Transient Stability Model Show Confirmation Dialog

If in the middle or at the end of a transient stability simulation the user tries to save the case, Simulator can prompt whether to save the case with the power system state after the transient stability simulation or to save the case with the power system state in place prior to starting the transient stability analysis. This option determines when the prompt will appear. If not prompted, the state will remain in the state following the transient stability simulation.

Automatic Update and Transfer Results to Power Flow Options

Specify the interval in time steps to update the displays or when to transfer results to the power flow model. Also, specify whether or not the displays should be updated and when/if results should be transferred to the power flow model. Any open plots will be refreshed automatically when performing the interval check while running transient stability.

Transient Stability Analysis Options: Power System Model

The Power System Model sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#). This contains options for how the actual transient stability numerical simulation is performed.

The screenshot shows the 'Options' dialog box with the 'Power System Model' sub-tab selected. The dialog has a title bar 'Options' and a note: 'Note: Changes made to option entries are saved immediately and will be applied during the next transient stability run.' The sub-tabs are: General, Power System Model (selected), Result Options, Generic Limit Monitors, Distributed Computing, and GIC Substation DC Voltages. The 'Power System Model' sub-tab contains several sections: 'Power System Values' with 'Nominal System Frequency (Hz)' set to 60.0 and 'System MVA Base' set to 100.00; 'Network Equations Solution Options' with 'Solution Tolerance (MVA)' set to 0.10000, 'Maximum Iterations' set to 25, 'Force Network Equation Update' set to 0.50, a checked 'Use Voltage Extrapolation' checkbox, and 'Inner Loop Mismatch Scalar' set to 1.0; 'Infinite Bus Modeling' with 'No infinite buses (recommended setting)' selected; 'Handling of Initial Limit Violations' with 'Modify Limits and Run' selected; 'Load Modeling' with 'Default Load Model' set to 'Constant Impedance', 'Minimum Per Unit Voltages for' set to 0.700 for Constant Power Models and 0.500 for Constant Current Models, and a 'Change' button; 'Exciter Saturation Model' with 'Quadratic (GE Approach)' selected; 'Integration Method' with 'Second Order Runge-Kutta' selected; 'Ignore Speed Effects in Generator Swing Equation' (unchecked); 'Governor Fast Valving' with a note 'Note: Needed for specific governor models such as TGOV3', 'Global Fast Valving Option' set to 'Initiate on Freq Deviation > (rad/sec)', and 'Fast Valving Parameter (rad/sec or sec)' set to 0.1.

Power System Values

Nominal System Frequency (Hz)

Set the nominal system frequency in hertz. The default is 60 Hz.

System MVA Base

MVA base for the system. The value cannot be changed here and is shown for informational purposes only. To change this value, go to the [Power Flow Solution General page](#) found under [Simulator Options](#).

Network Equation Solution Options

Solution Tolerance (MVA)

Specify the convergence tolerance used in the transient stability's power flow algorithm. This option setting only affects the transient stability power flow algorithm and does not affect the power flow model.

Maximum Iterations

Specify the maximum number of iterations allowed in the transient stability's power flow algorithm. This option setting only affects the transient stability power flow algorithm and does not affect the power flow model.

Force Network Equation Update

Specify a time in seconds which forces a full network equation update every so many seconds. This option can be helpful to avoid small unit oscillations caused by small mismatches.

Use Voltage Extrapolation

When this option is checked, an estimate of the voltage at time step will use the voltage at the previous three time steps to estimate what the next voltage will be. This essentially models the voltage as a quadratic function of time based on the last three time steps and creates an estimate of what the next voltage will be. This greatly aids in the initial guess of the network equation voltages at a time step and helps speed up the simulation.

Inner Loop Mismatch Scalar

PowerWorld Simulator uses a second order Runge-Kutta integration method. As a result network equation solutions are sometimes done at intermediate time steps as part of the integration method. Specifying a scalar here which is larger than 1.0 will allow those intermediate network equation solution to have a larger mismatch.

Infinite Bus Modeling

Select whether or not to use infinite buses. If using infinite buses, the power system slack bus(es) will be used as the infinite buses. If not using infinite buses, then an **Angle Reference** must be selected on the [Result Options](#). At infinite buses the angle does not change. When using infinite buses, these are used as the angle reference.

Default Load Model

Specify what load model to use by default in the transient stability simulation when the load does not have a load model characteristic defined. Normally it is best to define a load characteristic model specifically (it is even possible to define a load characteristic model which applies to the entire case), but if one is not specified this default will be used.

Minimum Per Unit Voltages for

When modeling loads as **Constant Power Models** or **Constant Current Models**, the load will start to fall off by a particular function if the voltage falls below the specified value. The values specified here are the same as the values for minimum voltages specified on the [Power Flow Solution Advanced Options page](#) found under [Simulator Options](#). Clicking the **Change** button will open the Power Flow Solution Advanced Options page.

Ignore Speed Effects in Generator Swing Equation

The generator swing equation includes a term divides the Mechanical Power by the per unit speed of the generator. Checking this box will ignore this speed effect. Normally you should not ignore this effect.

Governor Fast Valving

A few specific governor models such as TGOV3 include an effect called Fast Valving. This option specifies at what time the fast valving option is initiated. After choosing when you would like it to initiate you then specify either a frequency deviation in rad/sec or a time in second as the parameter **Fast Valving Parameter (rad/sec or sec)**.

Exciter Saturation Model

Option to specify the type of saturation function to use for an exciter model. Choices are Quadratic (GE Approach), Scaled Quadratic, or Exponential. When loading a PSLF DYD file it defaults to the Quadratic exciter saturation function. When loading a PSS/E DYR file it defaults to the Scaled Quadratic exciter saturation function. When loading a IPF SWI file it defaults to the Exponential exciter saturation function.

Integration Method

Option to specify whether to use the Second Order Runge-Kutta Order 2 (RK2) or Euler's method for the integration time step.

Transient Stability Dialog Options: Result Options

The Result Options sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#). The options on the Result Options determine how the results are stored during the simulation.

The screenshot shows the 'Options' dialog box with the 'Result Options' sub-tab selected. At the top, a note states: 'Note: Changes made to option entries are saved immediately and will be applied during the next transient stability run.' Below this are four tabs: 'General', 'Power System Model', 'Result Options' (which is active), and 'Generic Limit Monitors'. The 'Time to Begin Checking for Minimum and Maximum Results' section has three radio buttons: 'After last event' (selected), 'Immediately', and 'Custom Time' (with a value of 0.000). The 'Angle Reference Options' section has a group box 'Angle Reference' with five radio buttons: 'Average of Generator Angles' (selected), 'Weighted Average of Generator Angles', 'Specified Angle Reference Generator Terminal Angle', 'Specified Angle Reference Generator Internal Angle', and 'Synchronous Reference Frame (No Angle Shift)'. Below this is a text field for 'Angle Reference Generator' containing '36410 (MORRO 4) #H' and a 'Find...' button. There is also a checkbox for 'Initialize with Reference Angle at Zero'. A note at the bottom states: 'Note: This reference is not used if infinite buses are modeled'.

Time to Begin Checking for Minimum and Maximum Results

Minimum and maximum values for several bus and generator parameters are automatically stored during the analysis. This option determines when the recording of these minimum and maximum values should begin.

If you choose *After last event*, then the recording of minimum and maximum results will start after the last transient contingency element is processed for the simulation. This is the most common option and is used to prevent the recording of very low voltages during a fault event from being recorded as the minimum result when you are really only concerned with how the voltages recover after the fault clears.

If you choose *Immediately*, then the minimum and maximum result recording will start at the beginning of the simulation.

If choosing to use a *Custom Time*, a field will appear that will allow entry of a specific time in seconds when recording should begin.

Angle Reference Options

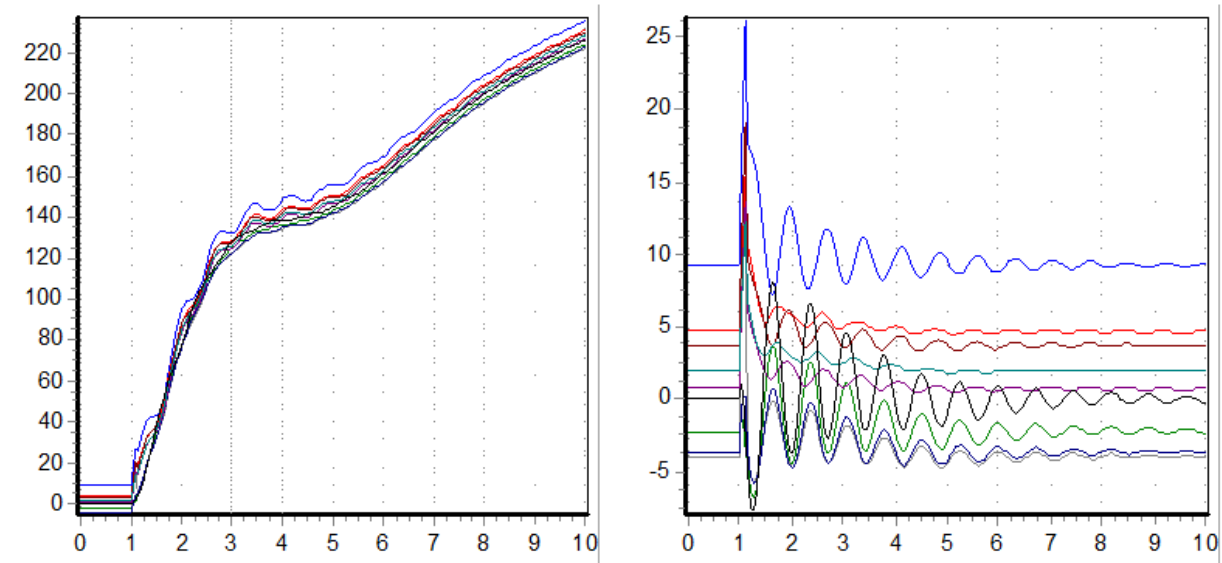
Note: if infinite buses are being modeled as part of the [Power System Model options](#), this option is not used and the fixed angle at infinite bus is automatically used as the reference.

Generator Rotor Angles in a transient stability simulation are calculated with respect to what is called the synchronous reference frame. The synchronous reference frame represents how the angles would move with respect to a theoretical rotating reference frame based on the system nominal frequency. When a transient stability simulation converges to a new steady state frequency which is

higher than nominal frequency, then inherently the rotor angles as expressed in this synchronous reference frame will continue to increase toward infinity. This will not indicate instability as long as the angles in relation to one another converge upon a new steady state.

The use of the synchronous reference leads to the angle reference option. This option specifies a reference which will be used to calculate an angle reference at each time step in a simulation. This angle reference will be calculated across the entire system. The generator field *Rotor Angle* recorded at a particular time in the results will then be equal to this system angle reference subtracted from the rotor angle on the synchronous reference frame. (Note: normally an angle reference option should be chosen because the generator rotor angle in the synchronous reference frame may be stored as an independent result variable called *Rotor Angle, No Shift*, which is distinct from the *Rotor Angle*.)

When looking at a plot of rotor angles in the synchronous reference frame, the plot may look as shown on the left below. If a angle reference option is chosen, then when plotting generator rotor angles, the plot will instead look as shown on the right. Both plots show the same system, but the plot on the right more clearly shows that the system is stable.



The options for angle reference are as follows

- *Average of Generator Angles* : reference equals the straight average of all rotor angles in the system
- *Weighted Average of Generator Angles* : reference equals the average of all rotor angles in the system weighted by each generators MVA base.
- *Specified Angle Reference Generator Terminal Angle*: the terminal bus angle of the generator specified on the dialog as the **Angle Reference Generator** will be used.
- *Specified Angle Reference Generator Internal Angle*: the internal bus angle of the generator specified on the dialog as the **Angle Reference Generator** will be used.
- *Synchronous Reference Frame (No Angle Shift)* : no reference angle is used so all rotor angles are reported on the synchronous reference frame.

Angle Reference Generator

This option must be set if the option for **Angle Reference Option** is set to one of the choices for using a specific generator.

Initialize with Reference Angle at Zero

This initializes the reference angle, regardless of the method selected for determining the reference angle, to zero at the beginning of the analysis. In practice this is done by recording the starting reference angle. At each time step the reference angle is determined and then this is offset by subtracting the original reference angle value from the current reference angle value and this new value becomes what is used as the reference angle at each time step.

Transient Stability Dialog Options: Generic Limit Monitors

The Generic Limit Monitors sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#). It contains the following special options apply generic limit monitors to all objects during the simulation.

Options

Note: Changes made to option entries are saved immediately and will be applied during the next transient stability run.

General Power System Model Result Options **Generic Limit Monitors** Distributed Computing GIC Substa

Synchronous Generator Limit Monitors

☒ Only Apply to Generators Without Relays

Monitor Type	Action to Take	Pickup Value	Pickup Time
Absolute Angle Deviation	Ignore	180.0 Deg	0.000 Seconds
Over Frequency	Ignore	62.40 Hz	2.000 Seconds
Under Frequency	Ignore	57.60 Hz	2.000 Seconds

Breaker Delay Time (Cycles) 0.0

Maximum Allowable Angle Difference (degrees) 1080.0

Synchronous Generator Limit Monitors

Only Apply to Generators Without Relays is a check-box which chooses to apply these generic limit monitors only to generators which do not have their own generator relay (such as a GP1) assigned to them.

There are three generic limit monitors which can be applied to all synchronous generators. These limit monitors are as follows

- **Absolute Angle Deviation** : monitors for synchronous generators whose rotor angle increases to a value which is substantially different than the initial rotor angle. This is a very crude attempt to monitor for out-of-step generators.
- **Over Frequency** : monitors for synchronous generators which experience high frequency.
- **Under Frequency** : monitors for synchronous generators which experience low frequency.

The limit monitors can then be configured to look for a violation of a specified Pickup Value that occurs for a specified Pickup Time. If a violation occurs you then specify an action to take due to this violation which can be either *Ignore*, *Log Warning*, *Trip (Open)*, or *Abort*. *Ignore* does nothing. *Log Warning* will return an Event in the Transient Stability Results indicating that a violation occurred and when it occurred. *Trip (Open)* will cause the generator to trip off-line (an event will also be logged stating this). If *Trip (Open)* is chosen then the generator will trip after a delay in cycles specified by the **Breaker Delay Time (cycles)**. Finally, *Abort* will causing the simulation to immediately abort when the limit monitor violation occurs.

For more customizable limit monitors, see the [Transient Limit Monitors](#) section.

Maximum Allowable Angle Difference (degrees)

This parameter determines the maximum allowable angle difference between any two generator rotor angles in the case. If this value is exceeded, the transient stability analysis will stop

Transient Stability Dialog Options: Distributed Computing

The Distributed Transient Stability Analysis tool is available as an add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for more details.

The Distributed Computing sub-tab is found on the [Options](#) page of the [Transient Stability Analysis dialog](#).

Distributed Computing is available for use with Transient Stability Analysis. In order to use distributed computing you must first configure a list of remote computers which can be utilized along with appropriate authentication information for those computers. The computer list and authentication information is common to all the distributed computing tools in Simulator and can be found in the [Simulator Options Dialog](#), or reached with the Distributed Computing Options button in the Distributed Computing sub-tab. They are described in [Distributed Computing Options](#).

The only option specific to Transient Stability Analysis for Distributed Computing is the following

Use Distributed Computing

Check this box to signify that when processing contingencies distributed computing should be used.

Transient Stability Analysis: Results Storage

The Results Storage page is found on the [Transient Stability Analysis dialog](#). The settings on this page determine

- What results are stored during the stability simulation.
- Where results are stored: RAM, Hard Drive, Both, or Neither.
- The frequency with which results are stored.

You may want to store results to RAM so that they can be viewed in the [Transient Stability Results: Time Values](#). Be careful however because if too many results are stored you will run out of RAM causing the simulation to fail or generally slow down your computer.

When storing results to Hard Drive you can be much more free with how much data is saved. After running you will still be able to load those results into RAM later (using the **Load from Hard Drive File into RAM results specified by Store to RAM Options** button, and you will also be able to [visualize Plot Charts](#) directly from the hard drive results.

When using the [Transient Limit Monitors](#), it can be reasonable to run a simulation which does not store any numeric results of the simulation. This is because when the goal is only to verify that all system performance criteria as described by the transient limit monitors are met, then the actual numerical results are not necessary.

Store Results to RAM

Check this box to store results using the [Save to RAM](#) Options.

Store Results to Hard Drive

Check this box to store results using the [Save to Hard Drive](#) Options.

Save the Results stored in RAM in the PWB file

The user can select to just store the transient stability parameters, store the parameters results, or not store either the transient stability parameters or results in the *.pwb file. Storing transient stability information in the *.pwb file makes it convenient to open a case and run the transient stability analysis, but storing this information, especially the results, could significantly increase the size of the *.pwb file. Auxiliary files can also be used for storing the parameters and results instead of, or in addition to, storing this information in the *.pwb file.

Save Results Every n Time Steps

When writing results to RAM or the Hard Drive, specify the number of time steps after which results should be stored. If a value of 10 is entered then results will only be stored every 10 time steps.

Note: this setting will NOT effect the use of [Generic Limit Monitors](#) or [Transient Limit Monitors](#). The Generic Limit Monitors and Transient Limit Monitors are an integrated part of the numerical integration solution and are checked at every time step regardless of how frequently results are saved to RAM or the Hard Drive.

Load from Hard Drive File into RAM results specified by Store to RAM Options

Click this button to cause Simulator to go out to the file specified in Save to Hard Drive Options and load in the values specified by the Store to RAM options into RAM. On the [Transient Stability Results: Time Values](#), only results stored in RAM can be viewed in the case information displays.

Transient Stability: Results Storage: Storage to RAM

The Results Storage page is found on the [Transient Stability Analysis dialog](#).

This section describes how to choose which results to store to RAM during the transient stability simulation. You may want to store results to RAM so that they can be viewed in the [Transient Stability Results: Time Values](#). Be careful however because if too many results are stored you will run out of RAM causing the simulation to fail or generally slow down your computer. The Store to RAM Options mostly consist of a list of all the various objects in the model and a list of field which can be stored during the transient simulation. This is depicted in the figure below. Any information needed for a subplot's horizontal axis data is automatically stored to RAM.

Result Storage

Where to Save/Store Results

☒ Store Results to RAM

☐ Save Results to Hard Drive

Save Results Every n Timesteps: 1

Load from Hard Drive File into RAM results specified by Store to RAM Options

☒ Save the Results stored to RAM in the PWB file

Store to RAM Options | Save to Hard Drive Options

Note: All fields that are specified in a plot series of defined plot will also be stored to RAM.

☐ Store Results for Open Devices | Set All to NO for All Types | Set Save All by Type ...

Generator | Bus | Load | Branch | DC Transmission Line | Multi-Terminal DC Converter | Area | Zone

Set All NO

Records | Geo | Set | Columns | Options

From Selection:

Make Plot

Make Plot Group by Field

Make Plot Group by Object

	Number of Bus	Name of Bus	ID	Area Name of Gen	Save All	Rotor Angle	Rotor Angle, No Shift	Speed	Mech Input	MW Terminal	Mvar Terminal	Term. PU	Field Voltage (pu)	Field Current	Stabilizer Vs	States of Machine	States of Exciter	S
1	10246	PERSONG1	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
2	10261	REEVE_G1	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
3	10262	REEVE_G2	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
4	10263	REEVE_G3	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
5	10318	SJUAN_G1	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
6	10319	SJUAN_G2	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
7	10320	SJUAN_G3	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
8	10321	SJUAN_G4	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
9	10394	LEF_G1	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
10	10395	LEF_G2	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N
11	10396	LEF_S1	1	NEW MEXICO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	NO	N

Tables of Devices

Results for Generators, Buses, Loads, Switched Shunts, Branches, DC Lines, Multi-Terminal DC Records, Multi-Terminal DC Converters, Areas, Zones, Interfaces, and Injection Groups can be stored during the transient stability analysis. Each type of object has a list of transient stability related parameters that can be stored. Toggling the appropriate field to **YES** for an object will store that parameter for the selected object. Some objects such as generators also have dynamic models associated with them for which dynamic states can be stored. For these you will see columns for **Machine States**, **Exciter States**, **Governor States**, etc... which can be set to **YES** to store those states. Each object also has a **Save All** field. If this field is set to **YES** for a particular object, all transient stability related fields (except dynamic states) will be stored for that particular object without having to set all of the fields to **YES** for all of the parameters. For dynamic states you can not use the Save All field and must toggle the specific state field instead.

Also note that any values which are set to be part of a plot series in a plot definition will automatically be stored to RAM

All results are presently stored in memory. This means that a memory limitation could be reached is storing a large number of results.

Save Results for Open Devices

Check this box to save results for devices that are currently open. It is possible that a device that is currently open could be closed during the transient stability analysis so this device should not be excluded from the results.

Set All NO

Click the **Set All NO** button to reset all fields regarding RAM storage for the present tab being viewed to *NO*.

Set All NO for All Types

Click the **Set All NO for All Types** button to reset all fields regarding RAM storage to *NO* for all object types.

Set Save All by Type...

Click this button to bring up a dialog containing check-boxes which allows you to choose which device types to toggle the **Save All** parameter to *YES* for. There will also be check-boxes regarding the various dynamic model state fields.

Plotting

Transient stability results are usually interpreted with the assistance of some sort of plot. There can be a large amount of data to sort through and options on this page as well as the [Plots](#) page are designed to assist in setting up meaningful plots. Plot Definitions must be created in order to view a plot. The plot buttons contained on the Results to Save page assist in creating Plot Definitions outside of the Plots page. See the [Plots](#) page for more detailed information about Plot Definitions and plot parameters. Once options on the Results to Save page are used to create plots, the plot definitions will show up on the [Plots](#) page.

In order to make a plot, objects and fields to include in the plot must be specified. How many objects and fields are selected will determine which of the following buttons are enabled and how the plot will be designed. To select objects and fields for a plot, simply select cells in the appropriate device table that include the fields and objects to show on a plot.

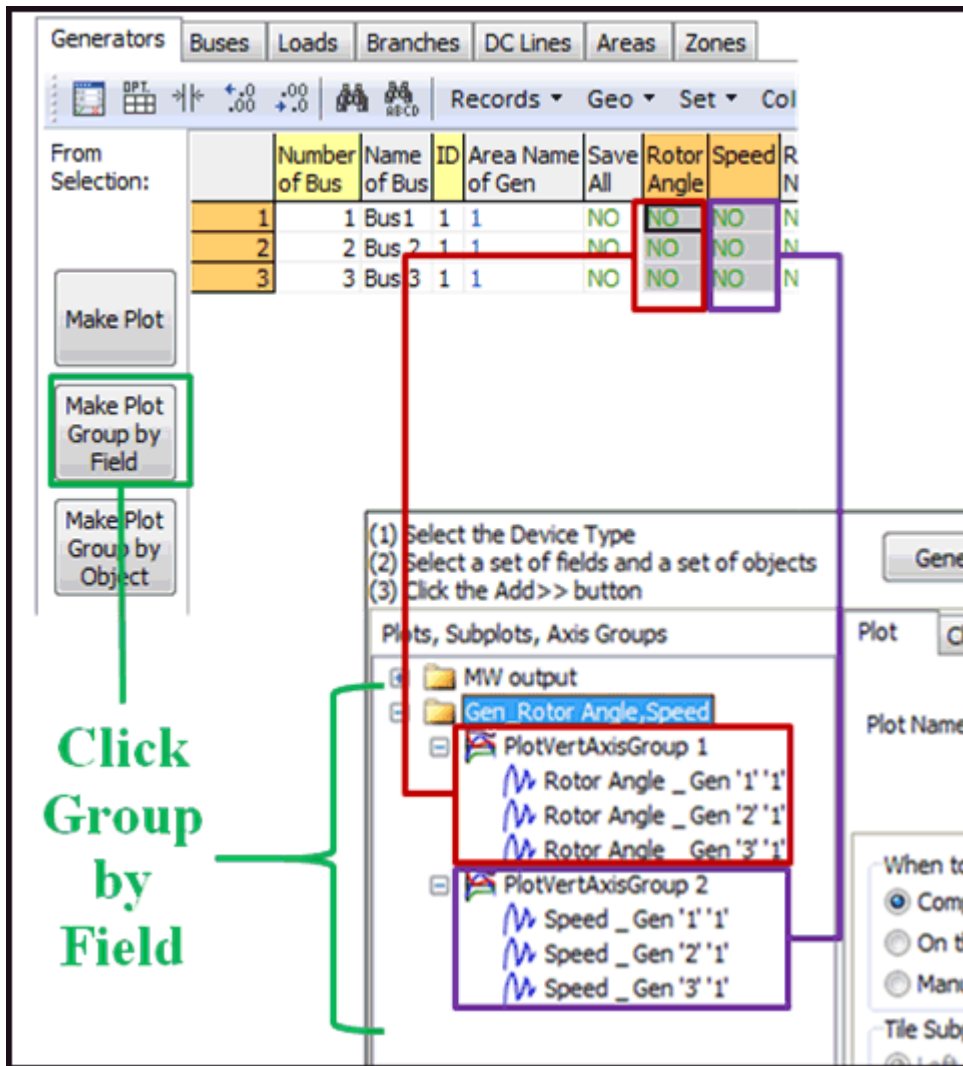
For example, to plot all generator rotor angles on a single plot, go to the Generators tab and select all of the records under the Rotor Angle field. After selecting the objects and fields, click the appropriate plot button, as described below, to create the plot definition.

Generator											
Bus Load Switched Shunt Branch DC Transmission Line Multi-Terminal DC Record Multi- >											
Set All NO											
Records Geo Set Columns >											
From Selection:	Number of Bus	Name of Bus	ID	Area Name of Gen	Save All	Rotor Angle	Rotor Angle, No Shift	Speed	Mech Input	MW Terminal	A
	1	1 Bus1	1	1	YES		NO	NO	NO	NO	NC
	2	2 Bus 2	1	1	YES		NO	NO	NO	NO	NC
	3	3 Bus 3	1	1	YES		NO	NO	NO	NO	NC
Make Plot											
Make Plot Group by Field											
Make Plot Group by Object											

As another example, to plot only the rotor angles and terminal MW value of only generators 1 and 2, go to the Generators tab. Move the MW Terminal field so that it is next to the Rotor Angle field. This can be done by using the [Display/Column Options dialog](#) or holding down the CTRL key and dragging a column to move it to a different place in the table. Select the records for generator 1 and

2 under both the Rotor Angle and MW Terminal fields. After selecting the objects and fields, click the appropriate plot button, as described below, to create the plot definition.

[illegible]



Make Plot

This button is enabled when at least one object record for a transient stability field is selected. This option will create a plot definition with a single subplot with all selected objects and fields appearing on the same axis group.

Make Plot Group by Field

This button is enabled when more than one transient stability field is selected. This option will create a plot definition with a single subplot with a separate axis group for each of the selected fields. All selected objects will appear on a given axis group.

Make Plot Group by Object

This button is enabled when more than one transient stability object is selected. This option will create a plot definition with a single subplot with a separate axis group for each of the selected objects. All selected fields will appear on a given axis group.

Transient Stability: Results Storage: Storage to Hard Drive

The Results Storage page is found on the [Transient Stability Analysis dialog](#).

This section describes how to choose which results to store to Hard Drive during the transient stability simulation. Results which are stored to Hard Drive can still be directly visualized using Simulator's [Plotting tools](#). The options are depicted in the figure below.

Result Storage

Where to Save/Store Results

☐ Store Results to RAM

☒ Save Results to Hard Drive

Save Results Every n Timesteps: 1

Load from Hard Drive File into RAM results specified by Store to RAM Options

☐ Save the Results stored to RAM in the PWB file

Store to RAM Options Save to Hard Drive Options

Directory in which to store all results TSR files for all contingencies

Location C:\temp Browse...

For each transient contingency, one file will be written to this directory.
The filename will be the name of the contingency with the *.TSR file extension.
Note: If no directory is specified, then the director of the case file will be used.
Note: Using a directory on your local computer is recommended.

Object Types to Include

☒ Generators

☒ Buses

☐ Loads

☒ Switched Shunts

☐ Branches

☒ DC Lines

☒ Multi-Terminal DC

☒ MTDC Converters

☐ Areas

☐ Zones

☐ Interfaces

☐ Injection Groups

☐ Only store every result for objects which meet the Area/Zone Filters

Edit Area/Zone Filters

TSR File Archiving

☐ Enable Auto-Archive of TSR Files

Maximum Number of Archive Files 6

Location

Specify the directory on your computer in which to store the results. For each [transient contingency](#), one file will be written to this directory. The filename will be the name of the contingency with the *.TSR file extension. When a transient stability simulation is started for a particular transient contingency, if a file already exists in this directory with the appropriate name then the file will be emptied and replaced with the results from the new stability run.

If this field is left blank, Simulator will attempt to save data in the same directory the case was loaded from. If a directory is specified but does not exist, Simulator will give you the option to create that directory or abort the run.

Because a large amount of data will be written to this file, it is best to not use a network drive which may cause some latency in writing results during the simulation. This latency can slow down the numerical simulation if the network drive is slow.

Object Types to Include

A list of objects types for which transient stability results can be stored. All fields (including the dynamic states) will be stored for each object in the TSR file. The only filtering allowed presently is to limit the results by the [Area/Zone/Owner filters](#) by checking the box **Only store every result for objects which meet the Area/Zone Filters**. Clicking the Edit Area/Zone Filters will open the [Area/Zone/Owner filters](#) case information display.

Transient Stability Analysis: Plots

The Plots page is found on the [Transient Stability Analysis dialog](#). The Plots page is broken into two sub-tabs that provide the means for creating and modifying plots:

Plot Designer

This page is used to create and modify plots.

Plot Definition Grids

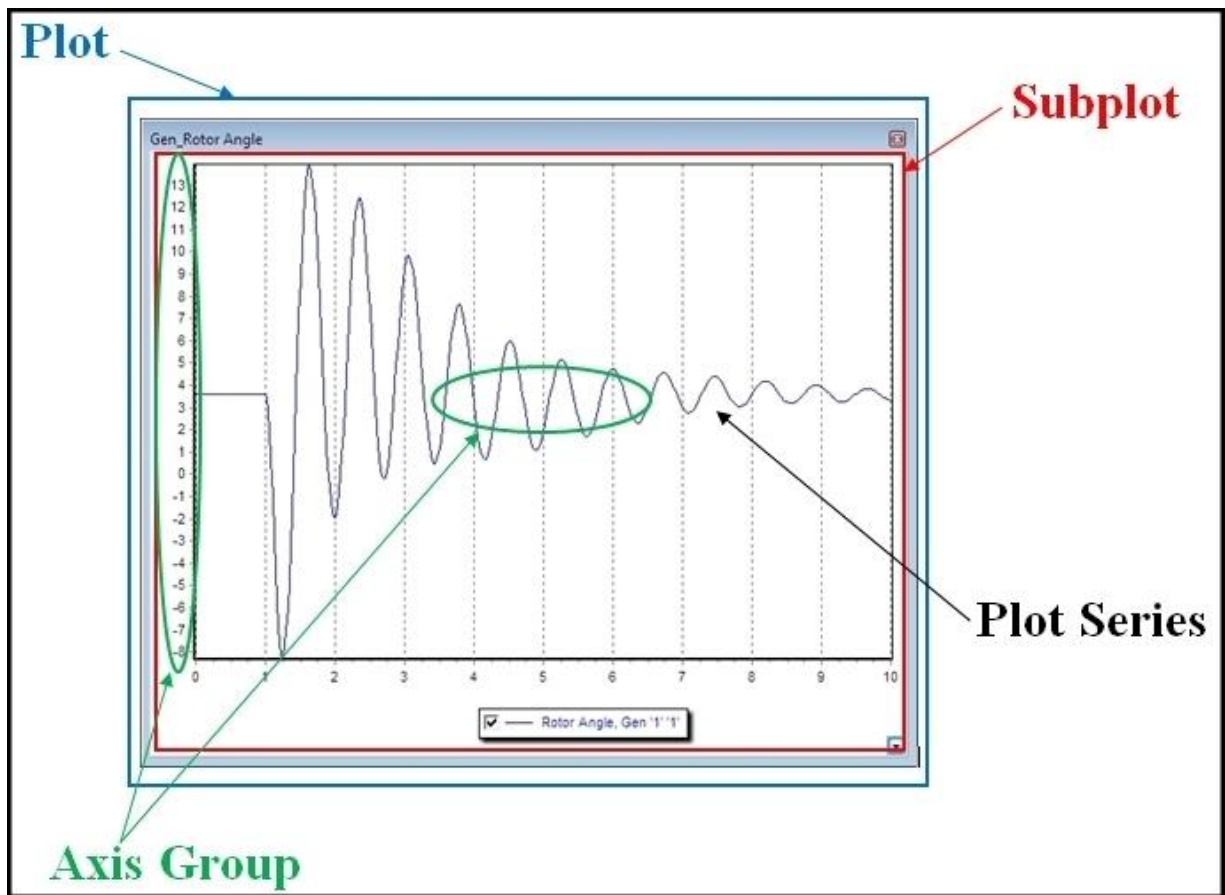
This page provides summary case information displays of the various components comprising a plot: Plots, Sub Plots, Axis Groups, and Plot Series. Once plots have been created using the [Plot Designer](#), they can be modified through the Plot Definition Grids. These grids also provide the format in which plots are stored in auxiliary files.

Once you have created a plot see the [Transient Stability Dialog: User Interaction with Plots](#) for many ways to interact with that plot to better view the results.

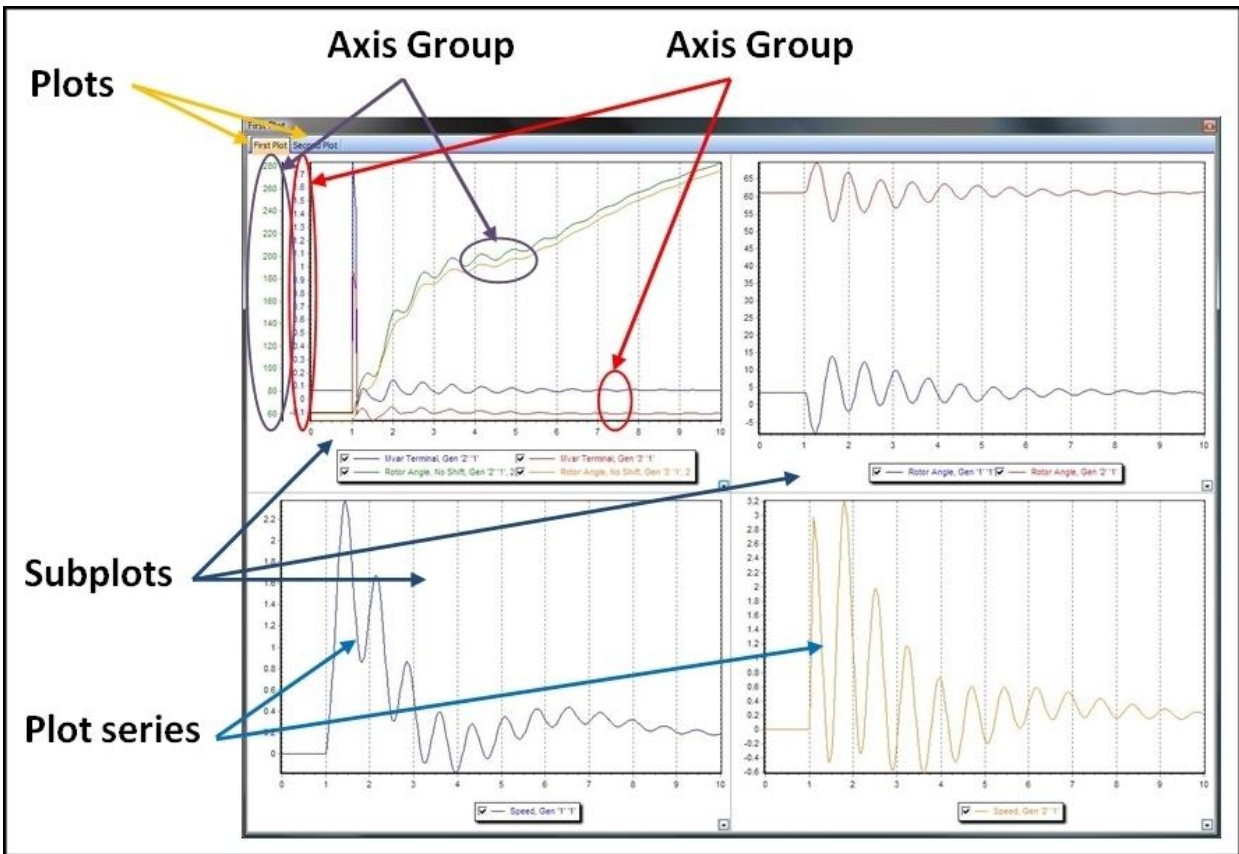
Plotting of transient stability results is an integral part of a transient stability tool. It is a primary mechanism through which transient stability results are viewed. Simulator has been designed to give a great deal of flexibility in the design of your plots allowing for multiple charts on a single plot as well as multiple vertical axes on a single chart. To discuss this in detail let us first define four objects which make up a plot definition

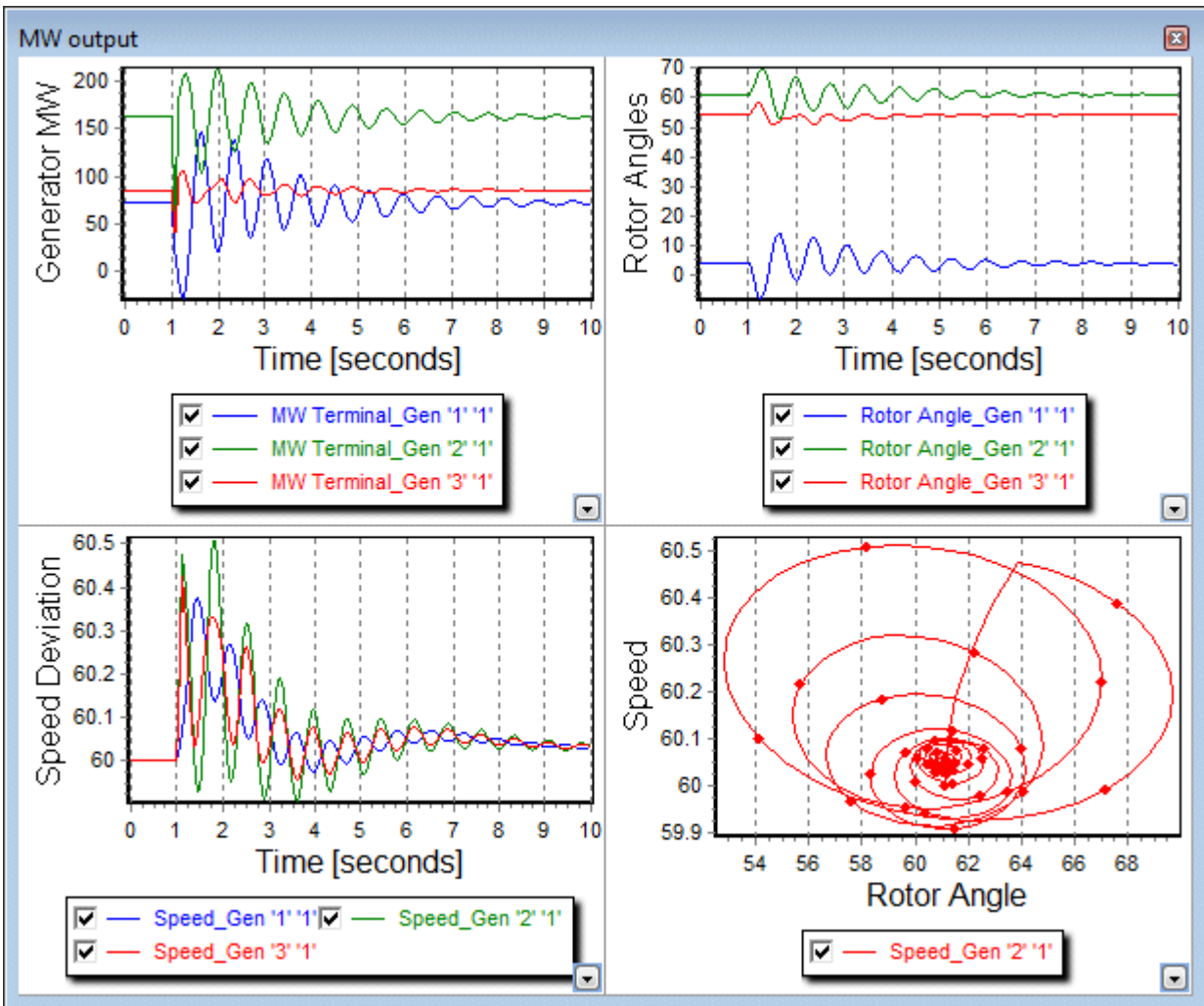
- *Plot* : a single window showing plotted results which may consist of 1 or more subplots
- *Subplot*: A subplot represents the actual graphical chart. It contains a single horizontal axis which is shared by all plot series contained in the subplot. The subplot contains one or more axis groups which represent vertical axes. For many plots there will be only one subplot
- *Axis Group* : A single axis group may contain many plot series. A single axis group represents a vertical axis which is shared by all the plot series it contains. For many subplots there will be only one axis group.
- *Plot Series* : A single plot series is a graphical line representing the trace of one numeric data series. For instance it may be the frequency of a particular bus.

As a simple example, consider the figure on the first figure below showing a plot that contains one subplot, that contains 1 axis group, that contains 1 plot series. Plots may also be much more complex as shown by the second figure below which has multiple subplots and multiple axis groups. Finally, the bottom-right subplot of the third example shows a subplot using a horizontal axis which is different than Time and thus shows a state space plot.



A more complicated example consisting of multiple plots, subplots, axis groups, and plot series is shown below:

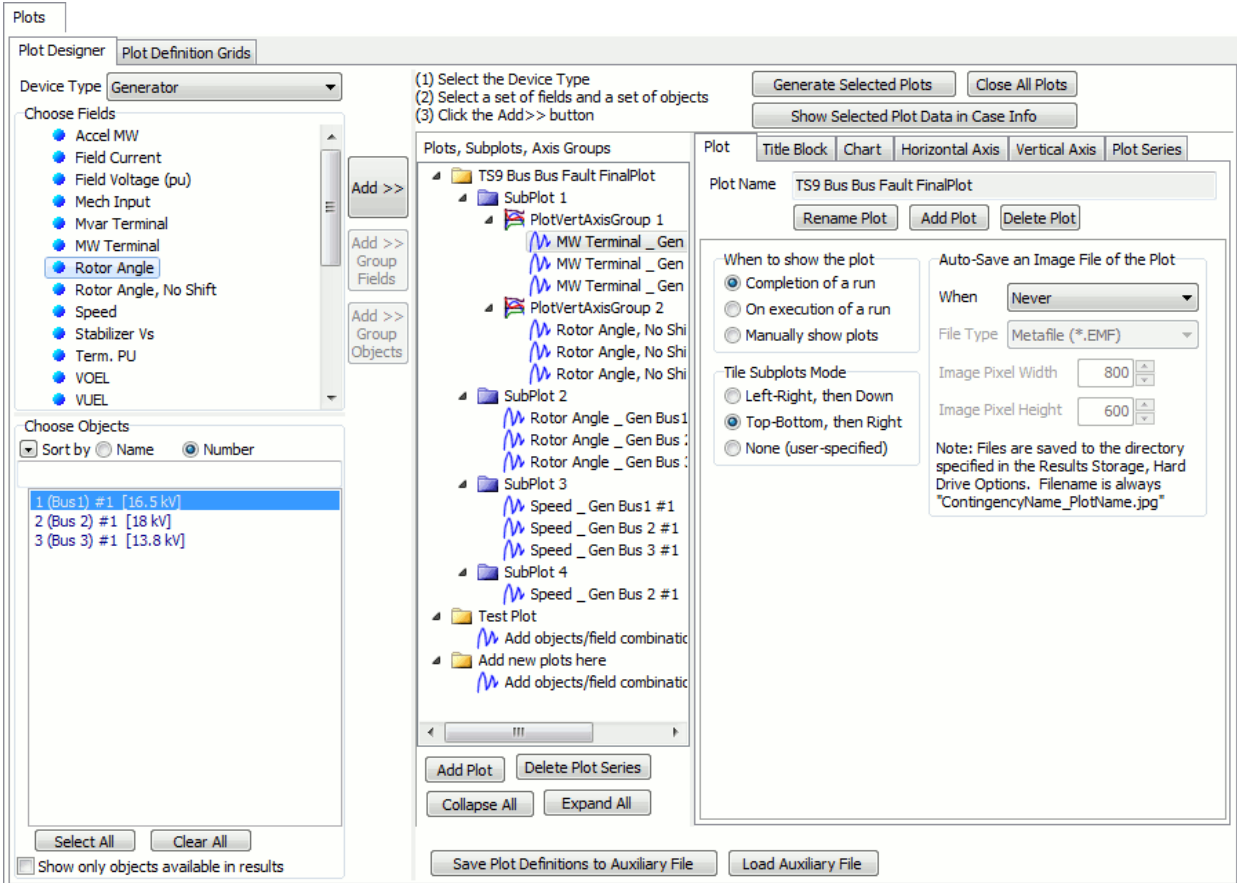




Transient Stability Analysis Plots: Plot Designer

The Plot Designer sub-tab is found on [Plots](#) page of the [Transient Stability Analysis dialog](#). Settings on this page determine which results will be plotted and when the plots will be made visible during the transient stability analysis.

The Plot Designer page is broken into three main sections. The left side of the page allows for the selection of devices and fields that should be included in a plot. The middle section defines the plots that are created and how these should be defined in terms of subplots and axis groups. The right section of the page allows modification of all of the plot components. The dialog is shown below.



Creating a New Plot

A new plot can be created in a number of ways:

- If no existing plot exists, simply selecting a **Device Type**, **Choose Fields**, and **Choose Objects** and then clicking the **Add >>**, **Add >> Group Fields**, or **Add >> Group Objects** buttons will create a new plot with a default name. (You may also drag a selection of fields over to the middle section listing the plot components to create a new plot).
- Click the **Add Plot** button found under the **Plots, Subplots, Axis Groups** list. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.
- Right-click on the **Plots, Subplots, Axis Groups** list and select Add Plot. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.

- Click the **Add Plot** button found on the **Plot** sub-tab that is part of the **Plots, Subplots, Axis Groups** section. A prompt will appear asking for the name of the plot. Enter the name and click OK. Then use the **Device Type**, **Choose Fields**, and **Choose Objects** options and the **Add >>** buttons to add data to a plot.
- Select fields and objects in the [Results to Save](#) tables.
- From the list of [Transient Limit Monitor Violations](#).

Right Section: Editing of Plot Definition

Most of the topic below discusses the creating of plot components. After plot components are created, selecting a particular plot component (plot, subplot, axis group, plot series) from the list in the middle of this portion of the dialog will change the sub-tabs that are available to the right of the list. Each of these sub-tabs allows for modifications of the different plot components as well as the components in which a particular component is contained, e.g. selecting a plot series will allow for modification of that particular series as well as the axis group in which the series is contained, the subplot in which the axis group is contained, and the plot in which the subplot is contained. Options can be changed for a plot before the plot is generated or while the plot is showing. Changing any of the options on the sub-tabs described below will update the current plot. Possible sub-tabs are as follows:

- [Plot](#)
- [Chart](#)
- [Horizontal Axis](#)
- [Vertical Axis](#)
- [Plot Series List](#)
- [Plot Series](#)

Device Type

Select the device type from the drop-down of all allowed types. Changing the device type will update the **Choose Fields** and **Choose Objects** accordingly.

Choose Fields

Choose the fields to be added to a plot. Multiple fields can be selected by holding down the CTRL key and selecting individual fields with the mouse, or holding down the Shift key and selecting the beginning and ending of a range with the mouse. You may also left-click and drag a list of fields here over to the middle section showing a list of plot components.

Dynamic States, Other Fields and Inputs

At the top of the field list is a list of the common fields which can be saved as part of the [Results Storage](#). For some types of objects, below this are several folders showing either Dynamic Model Input Fields, Dynamic Model Other Fields, or Dynamic Model States. The states represent the actually states of the dynamic equations of the mode. Other field represents other values of interest for the object which are not actually a state variable. Finally the input fields represent special input values to the dynamic model. Examples of input fields are the governor setpoint (Pref) and the exciter setpoint (Vref).

The entries underneath these folders will change as you select different objects from the **Choose Objects** pane. By default the entries will be very generic saying "Input 1", "Other 1", "State 1", "State 2", etc... If the selected object has a dynamic model associated with it then the entries will change to reflect this. If multiple objects are selected then the entries will only change if all objects selected have the same name for the specified State, Input, or Other field.

Choose Objects

Choose the objects to be added to a plot. Multiple fields can be selected by dragging the mouse down a selection, holding down the CTRL key and selecting individual fields with the mouse. The font colors of the objects lists in this list indicate whether any results are available for plotting presently. Black text indicates that there are no results available. Green text indicates that values are stored in RAM for this object. Blue text indicates that values are stored on the Hard Drive for this object.

Add >>

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one field and one object are selected:

Plot selected that contains only one subplot and one axis group - plot series for each field and object will be added to the selected plot in the single axis group and subplot

Subplot selected that contains only one axis group - plot series for each field and object will be added to the selected subplot in the single axis group

Axis group selected - plot series for each field and object will be added to the selected axis group

In any of these case if a plot series already exists for the selected field and object, a new plot series will not be added.

Add >> Group Fields

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one object and more than one field are selected:

Plot selected that contains only one subplot - axis groups will be created for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group

Subplot selected - axis groups will be created in the selected subplot for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group

Axis group selected - axis groups will be created in the subplot that contains the selected axis group. A new axis group will be created for each of the selected fields and plot series for each of the selected objects will be placed in each new axis group.

In any of these cases if an axis group for a selected field already exists in the corresponding subplot, a new axis group will not be created, but the plot series will be added to the existing axis group. If a plot series for a selected object already exists in the axis group, a new plot series will not be added. For an axis group to exist for a selected field, the axis group can only contain plot series for that particular field.

Add >> Group Objects

When clicked the selected fields and objects will be added to the appropriate plot component. Fields are selected from the **Choose Fields** list and objects are selected from the **Choose Objects** list. This button is enabled under the following conditions if at least one field and more than one object are selected:

Plot selected that contains only one subplot - axis groups will be created for each of the selected objects and a plot series for each of the selected fields will be placed in each new axis group

Subplot selected - axis groups will be created in the selected subplot for each of the selected objects and plot series for each of the selected fields will be place in each new axis group

Axis group selected - axis groups will be created in the subplot that contains the selected axis group. A new axis group will be created for each of the selected objects and plot series for each of the selected fields will be placed in each new axis group.

In any of these cases if an axis group for a selected object already exists in the corresponding subplot, a new axis group will not be created, but the plot series will be added to the existing axis group. If a plot series for a selected field already exists in the axis group, a new plot series will not be added. For an axis group to exist for a selected object, the axis group can only contain plot series for that particular object.

Generate Selected Plots

Click this button to generate plots for those selected in the **Plots, Subplot, Axis Groups** list. For a plot to be selected either the plot itself or one of its components needs to be selected. When generating a plot, an attempt will be made to retrieve from the results [stored to RAM](#). If results are not available for a particular plot series in RAM, then results will be retrieved from those values [stored to Hard Drive](#). If they are still not available then a plot will be generated which omits any plot series for which data can not be retrieved.

Close All Plots

Click this button to close all open plots.

Show Selected Plot Data in Case Info

Click this button to open a case info display

Save Plot Definitions to Auxiliary File

Click this button to save all of the plot definitions to an auxiliary file. This is useful when running analyses on power system models with common devices.

Plots, Subplots, Axis Groups

This is a listing of all plots that have been created along with associated subplots, axis groups, and data series. The list is arranged as a group of folders. Each plot is contained in its own folder. If there is more than one subplot for a plot, each subplot will be contained in a folder. Each subplot then contains a list of axis groups. If there is more than one axis group for a subplot, each axis group will be contained in a folder. Each axis group then contains a list of the plot series that it contains.

When adding devices and fields to a plot, the plot component selected in this list will determine where the data will be added when clicking one of the **Add >>** buttons. The **Add >>** buttons will be enabled according to the component selected in this list and the combination of devices and fields selected in **Choose Fields** and **Choose Objects**. Note you may also drag a list of fields from the **Choose Fields** portion of the dialog over to this listing to add new plots or add new plot series to the respective plot component.

There are several buttons located at the bottom of the plot listing. These buttons affect what and how things are shown in the list.

Add Plot

Click this button to add a new plot to the list.

Delete

The caption and functionality of this button will change depending on the particular type of plot component that is selected in the list. Clicking this button will then delete the selected component and any other components that are contained in the selected component. For example, if an axis group is selected and the Delete button is clicked, all plot series contained in the axis group will be deleted along with the axis group.

Note: you may also delete plot components by dragging them from this listing over to the **Choose Fields** pane.

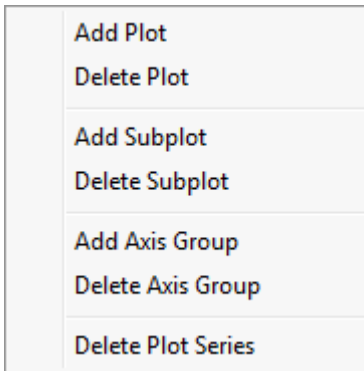
Collapse All

Click this button to collapse all folders in the list.

Expand All

Click this button to expand all folders in the list.

Right-clicking on components in the list will open a local menu that allows various options for adding or deleting plot components. The options that are available will change depending on the component selected.



Components can be moved from one plot to another or from one component to another by simply clicking and dragging the component to the new location. The new location must be a valid component that can contain the component being moved or the parent component for the new location must be able to contain the component being moved. For example, a subplot can be moved and dropped on an axis group with the subplot being placed in the plot containing the axis group.

Note: you may also delete plot components by dragging them from this listing over to the **Choose Fields** pane.

Transient Stability Analysis Plot Designer: Plot

The Plot sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the Plot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A plot is selected if any of its corresponding subplots, axis groups, or plot series are selected in the list.

The screenshot shows the 'Plot' sub-tab of the 'Plot Designer' dialog. At the top, there are five tabs: 'Plot', 'Title Block', 'Chart', 'Horizontal Axis', 'Vertical Axis', and 'Plot Series'. The 'Plot' tab is active. Below the tabs, the 'Plot Name' field contains 'TS9 Bus Bus Fault FinalPlot'. There are three buttons: 'Rename Plot', 'Add Plot', and 'Delete Plot'. Below these are two main sections. The left section, 'When to show the plot', has three radio buttons: 'Completion of a run' (selected), 'On execution of a run', and 'Manually show plots'. Below this is 'Tile Subplots Mode' with three radio buttons: 'Left-Right, then Down', 'Top-Bottom, then Right' (selected), and 'None (user-specified)'. The right section, 'Auto-Save an Image File of the Plot', has a 'When' dropdown set to 'Never', a 'File Type' dropdown set to 'Metafile (*.EMF)', and two numeric input fields for 'Image Pixel Width' (800) and 'Image Pixel Height' (600). A note at the bottom states: 'Note: Files are saved to the directory specified in the Results Storage, Hard Drive Options. Filename is always "ContingencyName_PlotName.jpg"'.

Plot Name

Name of the currently selected plot.

Rename Plot

Click this button to rename the currently selected plot. A dialog box will open in which the new plot name should be entered. Click OK on this dialog to change the plot name or Cancel to abandon the change.

Add Plot

Click this button to create a new plot. A dialog will open prompting for a name for the new plot. Click OK to create the new plot or Cancel to abandon the addition.

Delete Plot

Click this button to delete the currently selected plot.

When to show the plot

Set this option to determine when a plot should be shown. A plot can automatically be shown at the completion of a stability run or on execution of a stability run. If showing on execution, the plot will be updated as the analysis progresses. If set to be shown manually, the plot must be selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) page and the **Generate Selected Plots** button must be clicked in order to be shown. A plot can be shown manually in this manner at any time even if one of the options to show the plot automatically is selected.

Tile Subplots Mode

This option will only be enabled if a plot contains more than one subplot. This option dictates how the subplots will be placed within the plot window. If choosing **None (user-specified locations)**, the location of the subplot should be specified with the **Location** option found on the [Chart](#) sub-tab.

Auto-Save an Image of the Plot

Specify when and how plot images are saved to the Results Storage Hard Drive location. **When** defines when images are stored (can be *Never*, *After each contingency*, *Multiple Plot at the end*, or *Both*). **File Type** defines the format the image being saved (can be a *Metafile *.EMF*, *JPEG *.jpg*, *Bitmap *.bmp*, or a *GIF *.gif*). **Image Pixel Width/Height** specifies the dimensions of the saved image.

Transient Stability Analysis Plot Designer: Title Block

The Title Block sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the title block for a given plot.

Plot Name

Name of the currently selected plot.

Rename Plot

Click this button to rename the currently selected plot. A dialog box will open in which the new plot name should be entered. Click OK on this dialog to change the plot name or Cancel to abandon the change.

Add Plot

Click this button to create a new plot. A dialog will open prompting for a name for the new plot. Click OK to create the new plot or Cancel to abandon the addition.

Delete Plot

Click this button to delete the currently selected plot.

Show Title Block on Plot

Check this box to display the title block on the plot.

Show Date and Time in Title Block

Check this box to include the date and time in the title block on the plot.

Title Block Height %

This defines the portion of the overall window area dedicated to the title block.

Right Memo Width %

This defines the portion of the space dedicated to the title block memos taken up by the right memo. Note that the width this setting references is the combined width of the right and left memos, which is not necessarily the full width of the plot window.

Font Size

Set the font size of the memo text and date/time text here.

Where to show the title block

Set the location of the title block in the plot display window, either **Top of Plot** or **Bottom of Plot**

Left/Right Memos

These contain the text displayed in the memos to the left and right sides of the title block. Special strings may be entered, preceded by a '@' symbol, which will be automatically interpreted as an appropriate string to place in the memo.

@CTGName will display the appropriate contingency name

@CASENAME will show the name of the case presently open.

@BUILDDATE will show the Simulator patch build date. @DATETIME will show the present date and time.

@DATE will show the present date.

@TIME will show the present time.

Show Logo Image in Title Block

Check this box to display an image to the far left of the title block. You can specify an image with the "Logo Image" file browser, or leave the file path blank to use the PowerWorld logo.

Transient Stability Analysis Plot Designer: Chart

The Chart sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the subplot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A subplot is selected if any of its corresponding axis groups or plot series are selected in the list.

Plot Title Block **Chart** Horizontal Axis Vertical Axis Plot Series

Note: The chart attributes are assigned to one subplot.

Subplot Number 1 Add Subplot Delete Subplot

☒ Visible Background Color Change

Title

☒ Visible Font Size 12

☐ Specify Color Change

Footer

☒ Visible Font Size 12

☐ Specify Color Change

Location

Plot is set to automatically tile the subplots

Top 0.0

Left 0.0 Right 50.0

Bottom 50.0

Advanced Options Filename Browse... Clear

The Chart portion of the subplot is the background, header, and footer of the subplot.

Subplot Number

This is an informational field. Each subplot is assigned a number when it is created. The user cannot change this value.

Add Subplot

Click this button to add a subplot to the currently selected plot.

Delete Subplot

Click this button to delete the currently selected subplot.

Visible

Check this box to make the subplot visible when its corresponding plot is shown.

Background Color

Click the color box or the **Change** button to select a new color for the chart background.

Title

Check the **Visible** box to make a chart title shown. Check the **Specify Color** box and choose an desired color to specify a font color for the title. Specify the **Font Size** and the actual title text in the

box provided. Special strings may be entered, preceded by a '@' symbol, which will be automatically interpreted as an appropriate string to place in the title.

@CTGName will display the appropriate contingency name

@CASENAME will show the name of the case presently open.

@BUILDDATE will show the Simulator patch build date. @DATETIME will show the present date and time.

@DATE will show the present date.

@TIME will show the present time.

Footer

Check the **Visible** box to make a chart footer shown. Check the **Specify Color** box and choose an desired color to specify a font color for the footer. Specify the **Font Size** and the actual footer text in the box provided. Special strings may be entered, as described above.

Location

Set the **Top**, **Bottom**, **Left**, and **Right** location of the subplot within the plot window if choosing the option for user-specified subplot locations with the **Tile Subplots Mode** option found on the [Plot](#) sub-tab.

Advanced Options Filename

There are a number of advanced options associated with a chart that are only accessible through the local menu found on each subplot. This option allows a file containing these advanced options to be associated with the current subplot. If a file is specified, the advanced options contained in that file will be used when generating the subplot. Use the **Browse** button to browse for a file and use the **Clear** button to no longer associate a file with the current subplot. For more information about the advanced options, see the [Navigating Around a Plot, Advanced Options](#) section.

Transient Stability Analysis Plot Designer: Horizontal Axis

The Horizontal Axis sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the subplot that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. A subplot is selected if any of its corresponding axis groups or plot series are selected in the list.

The screenshot shows the 'Horizontal Axis' sub-tab of the 'Plot Designer' dialog. At the top, there are tabs for 'Plot', 'Title Block', 'Chart', 'Horizontal Axis' (selected), 'Vertical Axis', and 'Plot Series'. Below the tabs, a note states: 'Note: The horizontal axis is shared by all plot series on a subplot'. The 'Subplot Number' is set to '1', with 'Add Subplot' and 'Delete Subplot' buttons. The main area contains several sections: 'Show Horizontal Axis on Plot' (checked), 'Inverted' (unchecked), and 'Logarithmic' (unchecked). The 'Title' section has 'Visible' checked, 'Font Size' set to 12, and a 'Specify Color' button with a 'Change' button. The 'Scale' section has 'Automatic' checked for both 'Maximum' and 'Minimum' (both set to 0.00), and 'Round' unchecked. The 'Horizontal Axis Value' section has a 'Clear' button, 'Object plotted' set to an empty field with a 'Choose...' button, and 'Field plotted' set to 'Time' with a 'Choose...' button.

The horizontal axis is shared by all plot series on the selected subplot.

Subplot Number

This is an informational field. Each subplot is assigned a number when it is created. The user cannot change this value.

Add Subplot

Click this button to add a subplot to the currently selected plot.

Delete Subplot

Click this button to delete the currently selected subplot.

Show Horizontal Axis on Plot

Check this box to show the horizontal axis on the current subplot.

Inverted

Check this box to have the scale on the horizontal axis go from high to low values when moving left to right along the axis.

Logarithmic

Check this box to use a logarithmic scale on the horizontal axis of the current subplot.

Title

Check the **Visible** box to make a horizontal axis title shown. Check the **Specify Color** box and choose an desired color to specify a font color for the title. Specify the **Font Size** and the actual title text in the box provided. Special strings may be entered, preceded by a '@' symbol, which will be automatically interpreted as an appropriate string to place in the title.

@CTGName will display the appropriate contingency name

@CASENAME will show the name of the case presently open.

@BUILDDATE will show the Simulator patch build date. @DATETIME will show the present date and time.

@DATE will show the present date.

@TIME will show the present time.

Scale

Maximum

Check the box next to Automatic to have the maximum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the maximum value rounded when the Automatic checkbox is also checked.

Increment

Specify the increment between scale markings. If zero is specified, the increment will be determined automatically.

Minimum

Check the box next to Automatic to have the minimum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the minimum value rounded when the Automatic checkbox is also checked.

Horizontal Axis Value

The horizontal axis value is normally chosen to be time. This is the default if nothing else is selected. However, the option is allowed to choose a different value for the horizontal axis.

Clear

Click this button to restore the horizontal axis value to the default of Time.

Object Plotted

Select an object to be used for the horizontal axis values.

Field Plotted

Select the field for the associated object to be used for the horizontal axis values.

Transient Stability Analysis Plot Designer: Vertical Axis

The Vertical Axis sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the axis group that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab. An axis group is selected if any of its corresponding plot series are selected in the list.

The vertical axis is shared by all plot series in the selected axis group.

Axis Group Number

This is an informational field. Each axis group is assigned a number when it is created. The user cannot change this value.

Add Axis Group

Click this button to add an axis group to the currently selected subplot.

Delete Axis Group

Click this button to delete the currently selected axis group.

Show Vertical Axis on Plot

Check this box to show the vertical axis for this access group on the current subplot.

Inverted

Check this box to have the scale on the vertical axis go from high to low values when moving bottom to top along the axis.

Logarithmic

Check this box to use a logarithmic scale on the vertical axis of the current axis group.

Title

Check the **Visible** box to make the vertical axis title shown for the current axis group. Check the **Specify Color** box and choose an desired color to specify a font color for the title. Specify the **Font Size** and the actual title text in the box provided. Special strings may be entered, preceded by a '@' symbol, which will be automatically interpreted as an appropriate string to place in the title.

@CTGName will display the appropriate contingency name

@CASENAME will show the name of the case presently open.

@BUILDDATE will show the Simulator patch build date. @DATETIME will show the present date and time.

@DATE will show the present date.

@TIME will show the present time.

Scale

Maximum

Check the box next to Automatic to have the maximum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the maximum value rounded when the Automatic checkbox is also checked.

Increment

Specify the increment between scale markings. If zero is specified, the increment will be determined automatically.

Minimum

Check the box next to Automatic to have the minimum value on the scale automatically determined based on the results or uncheck this box and specify the value manually. Check the box next to Round to have the minimum value rounded when the Automatic checkbox is also checked.

Transient Stability Analysis Plot Designer: Plot Series List

The Plot Series List sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the plot series contained in the axis group that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab.

Plot | Title Block | Chart | Horizontal Axis | Vertical Axis | **Plot Series List**

Note: The vertical axis contains the list of plot series below

Axis Group Number

Records Set Columns

	Object	Variable	Style	Visible	Color	Thickness	Dashed	Stairs	Symbol Every	Symbol	Width	Height
1	Bus '55272'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
2	Bus '55274'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
3	Bus '55275'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
4	Bus '55278'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
5	Bus '55280'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
6	Bus '55281'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
7	Bus '55285'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
8	Bus '55289'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
9	Bus '55300'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
10	Bus '55301'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
11	Bus '55302'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
12	Bus '55303'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
13	Bus '55304'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
14	Bus '55310'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
15	Bus '55312'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
16	Bus '55313'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5
17	Bus '55314'	TSBusVPU	Line	YES		1	Solid	No	0	Diamond	5	5

Axis Group Number

This is an informational field. Each axis group is assigned a number when it is created. The user cannot change this value.

Add Axis Group

Click this button to add an axis group to the currently selected subplot.

Delete Axis Group

Click this button to delete the currently selected axis group.

Plot Series List Table

This is a [case information display](#) listing all of the plot series contained in the currently selected axis group. Common options like color, thickness, and style that determine how a plot series is shown are listed in this display and can be modified here. The same fields can be modified for a single selected plot series on the [Plot Series](#) sub-tab.

Transient Stability Analysis Plot Designer: Plot Series

The Plot Series sub-tab is available on the [Transient Stability Analysis dialog](#) on the [Plot Designer](#) sub-tab of the [Plots](#) page. This tab provides information about the plot series that is currently selected in the **Plots, Subplots, Axis Groups** list found on the [Plot Designer](#) sub-tab.

Plot | Title Block | Chart | Horizontal Axis | Vertical Axis | **Plot Series**

Note: The plot series represents ONE actual line on a chart

Object plotted: Gen '2' '1'

Field plotted: MW Terminal Delete Plot Series

☒ Plot Series Visible

Color: Change

Plot Series Type: Line Series

Line Attributes

Style: Solid

Thickness: 2

Stairs: No

Symbol Every: 0

Point Attributes

Style: Diamond

Height: 5

Width: 5

Object plotted

This shows the object that is represented by the selected plot series. This is for informational purposes only and cannot be changed here.

Field plotted

This shows the field that is represented by the selected plot series. This is for informational purposes only and cannot be changed here.

Delete Plot Series

Click this button to delete the selected plot series.

Plot Series Visible

Check this box for the plot series to be shown on the plot.

Color

Click the color box or the **Change** button to select the color of the line drawn for the plot series.

Plot Series Type

Select whether the plot series should be shown by a *Line Series* or a *Point Series*. If shown by a *Point Series*, the **Point Attributes** options will be enabled allowing selection of the style and size of the points.

Line Attributes

These options will only be enabled if the **Plot Series Type** is a *Line Series*.

Style

Solid, Dashed, Dot, Dash Dot, and Dash Dot Dot are the valid options for the line style.

Thickness

Set the thickness of the line.

Stairs

Select whether the line series should be drawn with No stairs, Stair, or Inverted stair. The stair options connected successive points in a stair pattern instead of a straight line.

Symbol Every

Set this option to something other than zero if a symbol should be placed on the line for better identification. Making this value larger will make the symbol placing less dense. Use the **Point Attributes** options to define the style and size of the symbols.

Point Attributes

These options will only be enabled if the **Plot Series Type** is a *Point Series* or choosing to include symbols on a *Line Series* by setting the **Symbol Every** option to something other than zero.

Style

This determines the shape of any plot series symbols.

Height

This sets the vertical size of the plot series symbols.

Width

This sets the horizontal size of the plot series symbols.

Transient Stability Dialog: User Interaction with Plots

The Plots page is found on the [Transient Stability Analysis dialog](#).

A set of common user-interaction features with the plot has been implemented. To Pan on a chart, simply right-click and drag on the plot. This will move the view up, down, left, or right depending on which direction you drag the mouse. To zoom in on the plot, left-click and drag towards to the left and form a square into which the view will zoom. To zoom out on plot left-click and drag towards the right and the view will zoom back out to the default zoomed out view.

Navigating Around a Plot

Pan Around Plot

Right-click on the background portion of a subplot and drag with the mouse.

Zoom In

Left-click on the background portion of a subplot and drag down with the mouse.

Zoom Out

Left-click on the background portion of a subplot and drag up with the mouse. The view will zoom back out to the default zoomed out view of the plot.

Data Information

Left-clicking on a plot series will provide information about the selected point. This will also be shown in a hint which appears as you hover your mouse over a particular plot series.

Making Plot Series Visible

Below each subplot chart there is a key listing all of the plot series on that subplot. Check and uncheck the box next to a plot series to make that plot series visible or not visible. (Note: after a chart has more than 12 plot series, the legend is not drawn by default.)

Chart Local Menu

A local menu is available for the chart by right-clicking below the chart portion of a subplot, right-clicking on a plot series, or clicking the little drop-down arrow found in the bottom right-hand corner of each subplot. (Note: Right-clicking on the region of the chart showing the plot series will not bring up this menu because right-clicking activates the dragging on these charts.) The choices which appear are as follows.

Print Preview...

Provides the ability to access a print options for the chart

Export...

Provides the ability to export an image of the chart to file types including an Enhanced Metafile, Bitmap, JPEG, PDF, SVG, PNG, VML, or EPS

Copy

Will copy the image as a metafile to the Windows clipboard so that the image can then be pasted into another program.

Export with Options

Provides access to export images, but in addition provides access to export the data behind the plot series to a text file, XML, HTML or Excel. To export data you must go to the **Data** tab on the dialog which appears.

Advanced Options

Remember that the most common options available for plots and associated components are discussed in the [Plot Designer](#) section.

There are a number of advanced options that are associated with a subplot. These advanced options are only accessible through a local menu available with each subplot. These advanced options can be saved or loaded from file.

Load Advanced Results

Choose this option to open a file which has the Advanced Options stored in it.

Save Advanced Result

Choose this option to save a file which has the Advanced Options stored in it.

Print All Charts and Title Block

Prints the current plot display

Copy All Charts and Title Block

Loads the current plot display onto the clipboard

Close All Plots

Closes all open plots.

Transient Stability Transient Limit Monitors

The Transient Limit Monitors page is found on the [Transient Stability Analysis dialog](#).

When running a transient stability simulation, the actual numerical results and plotting is useful, but sometimes what you really want to know is if and when any particular Operating Standards were violated. For example, in the Western United States there are particular standards for voltage dips and frequency deviations referred to as WECC Category B and WECC Category C standards. You can think of this similarly to how Limit Monitoring works in traditional power flow based contingency analysis. In traditional contingency analysis we do not report the voltage at every bus and the flow on every line during each contingency. Instead we only report violations of limits, which greatly reduces the amount of output data created by the tool.

In order to provide the ability to flag violations of these standards, PowerWorld has created an object called a *Transient Limit Monitor* which provides a great deal of flexibility for allowing the simulation to automatically monitor for these types of standards without requiring us to store the entire trace of each monitored quantity. When creating a transient stability monitor, you choose a field for a particular type of object and then build a description of what is considered a limit violation.

Defining Transient Limit Monitors

To define Transient Limit Monitors, look on the Transient Stability dialog under Transient Limit Monitors and choose the Transient Limit Monitors option. When choosing this a case information display listing the transient limit monitors will be shown. For more information see [Transient Stability : Defining Transient Limit Monitors](#).

Viewing Transient Limit Monitor Violations

While running a transient stability simulation, violations of the transient limit monitors defined will be recorded and listed under the Transient Limit Monitors\Monitor Violations table on the Transient Stability Analysis Dialog. Monitor Violations will be designated by which transient contingency caused them so that when [running multiple transient contingencies](#) you will be able to see which contingency caused the violation. It may be feasible for you to abandon storing all the transient stability numerical results and instead only store the violations of the limit monitors. If violations are found you can then revisit the run and store appropriate results. For more information on monitor violations see [Transient Stability : Transient Limit Monitors Violations](#).

Transient Stability : Defining Transient Limit Monitors

The Transient Limit Monitors page is found on the [Transient Stability Analysis dialog](#).

When inserting a new transient limit monitor you are presented with the following dialog. This gives you great flexibility in defining monitors for the various values which can be recorded in a transient stability run. Note that there are some very simple [Generic Limit Monitors](#) for synchronous machines which are define as part of the Transient Stability options that may also be of use.

The screenshot shows the 'Transient Limit Monitoring Dialog' window. At the top, the 'Name' field is set to 'WECC Category B Voltage Dip Load Bus'. Below this are buttons for 'Save', 'Save As', 'Rename', and 'Delete'. The 'Active' checkbox is checked. Under 'Objects and Field Monitored', 'Device Type' is 'Bus' and 'Field' is 'V (pu)'. The 'Filter objects for monitoring' section has radio buttons for 'All', 'Meets Filter', 'Load Buses Only' (selected), 'Non Load Buses Only', and 'Area/Zone Filters'. A 'Define Filter...' button is next to the 'Load Buses Only' option. The 'Action To Take' section has radio buttons for 'Log Violation Only' (selected), 'Abort Simulation', and 'Trip (Open) Device'. To the right, 'Use a time delay of' is set to '0.00' seconds, and 'Maximum number of violations of this limit monitor to store' is set to '100'. The 'Limit Value' is '-25.00 %' and 'Limit Duration' is '0.0000 Seconds'. 'Limit Duration Units' has radio buttons for 'Seconds' (selected) and 'Cycles'. The 'Limit Type' section has radio buttons for 'Lower' (selected) and 'Upper'. The 'Meaning of the values specified' section has radio buttons for 'Actual value', 'Percent of initial value', 'Deviation from initial value', and 'Percent deviation from initial value' (selected). There is an unchecked checkbox for 'ABS: After using meaning, take absolute value'. A 'Show Special Triggers to Start or Stop Transient Limit Monitoring' button is at the bottom. The dialog has 'OK', 'Help', and 'Cancel' buttons at the very bottom. A small graph on the right shows a waveform with a horizontal line for the 'Limit Value' and a red circle indicating a 'Violation' that lasts for a 'Limit Duration'.

In the top portion of the dialog you first give the transient limit monitor a name. There is then an option to make a transient limit monitor **Active** or not. Below the middle of the dialog there is the Action To Take options: **Log Violation Only**, **Abort Simulation** or **Trip (Open) Device**. The **Log Violation Only** will show the violation only in the message log. The **Abort Simulation** will abort simulation at a specified time after a violation of the monitor occurs. The **Trip (Open) Device** will option to trip/open a device" if a monitoring violation occurs (this works for Buses, AC Lines, Generators, Loads, and DC Lines). There is also a choice for **Maximum number of violations of this limit monitor** to store. This is needed so that too many limit violations of the same limit

monitor are not created. If you know the lowest 100 bus frequencies, you don't need any more than that.

There are then choices to specify the **device type** and a **filter** which together will determine all the objects which are ultimately monitored by the transient limit monitor. Note that a decision regarding which objects meet the filter will be done at the start of a stability simulation based on the steady state values of the power system. You then specify which **field** of the device type will be monitored.

The top portion of the dialog describes which particular object/field pair traces are going to be monitored. The bottom portion of the dialog will define what shape of the trace will be considered a limit violation. For the basic limit monitor you specify three values: **Limit Value**, **Limit Duration**, and **Limit Type**. These choices work together to determine what is considered a violation. Typically limit types are *Lower* and a trace will be considered violated if the value is below the Limit Value for a duration specified by Limit Duration. For a *Upper* limit type, then it must be above the Limit Value instead. The choice for **Meaning of the values specified** affect how the Limit Value and values specified for special triggers discussed shortly are interpreted. The first choice is Actual Value which simply means the limit value represents the exact numerical value considered a violation. The other choices are based on a deviation from the initial steady state value or a percent or percent deviation from the initial value. As an example, for a system with a 60 Hz nominal frequency, the following are equivalent: [Actual value = 59.6 Hz] and [Deviation from initial value = -0.4 Hz]. Similarly, when monitoring a voltage the following are equivalent: [Percent of initial value = 70%] and [Percent deviation from initial value = -30%]. Finally there is a option **ABS: After using meaning, take absolute value**, which will take the absolute value of the value.

By default the transient limit monitoring does not start until after the last user-specified contingency event occurs. This is done because values such as voltage will obviously be violating during a bus fault (they'll be zero!), so what you really want to know is whether the voltage is below a value for a specified duration after the fault clears. To change this default behavior, there are also special options available by clicking on the **Show Special Triggers to Start or Stop Transient Limit Monitoring** button which allow you to specify triggers for when the monitoring starts and stops. These are shown in the dialog below. The values specified here will obey the choice for **Meaning of the values** specified discussed above.

	Use	Value
Stop monitoring if a value is above	<input type="checkbox"/>	0.00 %
Start monitoring if a value is above	<input type="checkbox"/>	0.00 %
Stop monitoring if a value is below	<input type="checkbox"/>	0.00 %
Start monitoring if a value is below	<input type="checkbox"/>	0.00 %

Built-In Transient Limit Monitors for WECC Standards

Simulator has built in the ability to automatically insert transient limit monitors which represent WECC Category B and WECC Category C standards. This can be inserted going to the Transient Limit Monitors case information display choosing the **Records** menu dropdown from the case information

toolbar and then choosing the appropriate option for **Build WECC Category B Standard Monitors** or **Build WECC Category C Standard Monitors**. Note that by default the WECC Category B standard monitors are always inserted into a new power system case. The standards are initially marked as Active = NO however so they must be set to Active = YES in order to use them.

The WECC Category B Standards create four transient limit monitors.

- *WECC Category B Voltage Dip for Non-Load Buses* will monitor all non-load buses for any voltage dip of 30% below the initial voltage value at any time.
- *WECC Category B Voltage Dip for Load Bus* will monitor all load buses for a voltage dip of 25% below the initial voltage at any time.
- *WECC Category B Voltage Dip for Load Bus Duration* will monitor all load buses for a voltage dip of 20% below the initial voltage for a duration of 0.3333 seconds (20 cycles) and the dialog is shown above.
- *WECC Category B Frequency* will monitor all load buses for a frequency dip below 59.6 Hz for a duration of 0.10 seconds (6 cycles) and the dialog is shown to the right as an example.

The WECC Category C Standards create three transient limit monitors.

- *WECC Category C Voltage Dip Any Bus* will monitor all buses for a voltage dip of 30% below the initial voltage at any time.
- *WECC Category C Voltage Dip Any Bus Duration* will monitor all buses for a voltage dip of 20% below the initial voltage for a duration of 0.6667 seconds (40 cycles).
- *WECC Category C Frequency* will monitor all load buses for a frequency dip below 59.0 Hz for a duration of 0.10 seconds (6 cycles).

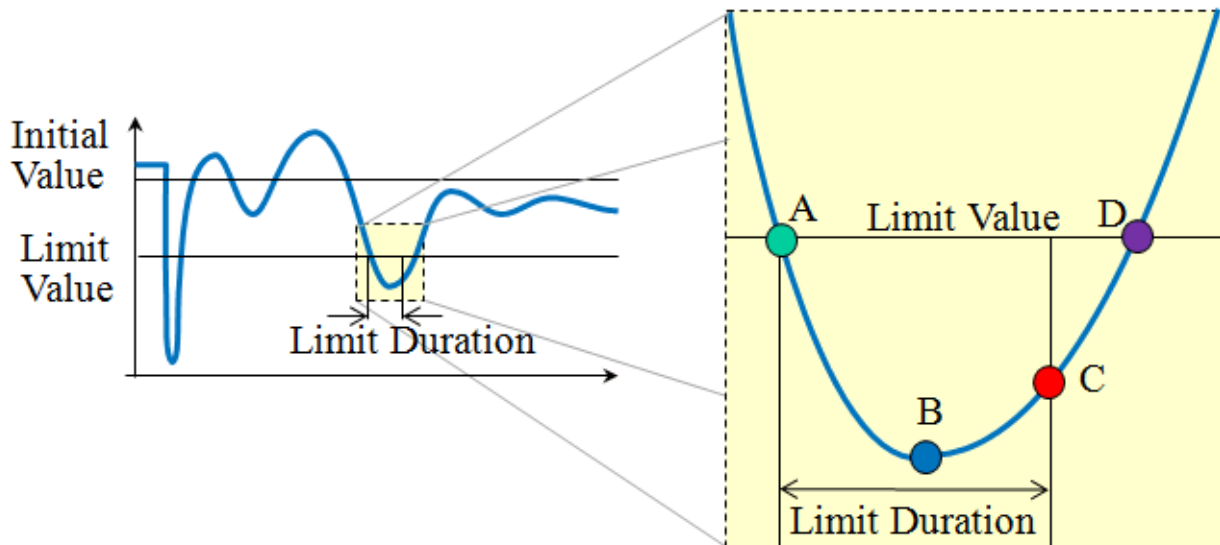
Transient Stability : Transient Limit Monitors Violations

The Transient Limit Monitors page is found on the [Transient Stability Analysis dialog](#).

As a transient stability run is processed, any violations of the Transient Limit Monitors will be stored as a monitor violation and can be seen in the Monitor Violations table. An example is shown in the following case information display. Note that there is an option in **the Records Menu** for this object to **Make a New Plot**. Choosing to make a new plot will automatically create a new plot definition on the [Plot Designer](#) portion of the dialog.

Transient Limit Monitors										
Monitor Violations										
	Limit Monitor Name	Contingency Name	Violated Device	Limit Monitor Variable	Initial Value	Actual Value	Converted Value	Time of Value	Actual Value Start	Converted Value Start
1	WECC Category B Voltage Dip Nc	My Transient Contingency3	Bus '2'	TSBusVPU	1.025000	0.570382	-44.352936	1.5083	0.570382	-44.352936
2	WECC Category B Voltage Dip Nc	My Transient Contingency3	Bus '3'	TSBusVPU	1.025002	0.034863	-96.598785	1.5083	0.034863	-96.598785
3	WECC Category B Voltage Dip Nc	My Transient Contingency3	Bus '7'	TSBusVPU	1.025773	0.510393	-50.243046	1.5083	0.510393	-50.243046
4	WECC Category B Voltage Dip Nc	My Transient Contingency3	Bus '9'	TSBusVPU	1.032357	0.202323	-80.401794	1.5083	0.202323	-80.401794
5	WECC Category B Voltage Dip Nc	My Transient Contingency3	Bus '4'	TSBusVPU	1.025792	0.711962	-30.593933	1.5250	0.711962	-30.593933
6	WECC Category B Voltage Dip Lo	My Transient Contingency3	Bus '5'	TSBusVPU	0.995640	0.631304	-36.593128	1.5083	0.631304	-36.593128
7	WECC Category B Voltage Dip Lo	My Transient Contingency3	Bus '6'	TSBusVPU	1.012659	0.534131	-47.254669	1.5083	0.534131	-47.254669
8	WECC Category B Voltage Dip Lo	My Transient Contingency3	Bus '8'	TSBusVPU	1.015889	0.377296	-62.860462	1.5083	0.377296	-62.860462
9	WECC Category B Voltage Dip Lo	My Transient Contingency3	Bus '8'	TSBusVPU	1.015889	0.171945	-83.074455	1.8417	0.377296	-62.860462

The key fields for the Monitor Violations will be the *Limit Monitor Name*, *Transient Contingency Name*, and the identity of the *violated device*. In addition to these keys, up to four additional points will be reported. Each of these four points will be represented by a *value* and a *time of value* representing the y-x point of the trace for that point. This gives up to 8 additional values. The four points are depicted in the following figure.



A description of the four points are as follows.

- Point A represents the initial time at which the trace has violated the Limit Value. When processing the transient stability run, a monitor violation will not be generated at the point but the trace will start to be tracked. Also note that as soon as the trace no longer violates the limit value, point A will essentially be reset.
- Once a Point A has been encountered and the trace is being tracked, a value/time pair for Point B will be maintained as part of the result to indicate when the most extreme violation of the limit

value occurs. Note that point B may occur before or after point C depending on the shape of the trace.

- Point C represents the time at which the trace completely violates the limit monitoring. At this point the limit value has been violated for the appropriate limit duration. Once this violation occurs then a Transient Limit Monitor Violation object will be generated for the results. Points A and B will be stored at this point as well, although point B may be modified if the trace continues to get worse. If the limit duration is zero, then point C and point A are the same point.
- After the violation occurs, the tracking of the trace will continue to see if at a later time during the simulation the value no longer violates the Limit Value. If this occurs then a Point D will also be stored with the monitor violation object. This provides information about how long the value was ultimately violated so that you know if you almost met the standard. For instance if you have a limit duration of 0.3333 seconds (20 cycles) you might want to know the difference between a violation that lasted 0.34 seconds and one that lasted 1.2 seconds.

Transient Stability Analysis: Results

The Results page is found on the [Transient Stability Analysis dialog](#). This page provides a summary of various results from a transient stability analysis run.

Results	
Time Values From RAM Minimum/Maximum Values Summary Events Solution Details	
Time Solution Started:	5/5/2010 3:13:10 PM
Time to Solve (Seconds)	89.342
Maximum Angle Difference (Deg)	292.207
Time of Max. Angle Diff. (Seconds)	5.000
Time Solving Differential Equations	0.000
Newton Solution Results	
Total Newton Solutions	996
Number of Jacobian Factorizations	19
Number of Forward/Backward Substitutions	1112

The Results page is divided into several sub-tabs:

Time Values from RAM

This page contains a table of all results selected with [Results to Store to RAM](#). For details on this display see [Transient Stability Results: Time Values from RAM](#).

Minimum/Maximum Values

The Minimum/Maximum Values tab contains two lists: one of generators and one of buses. Each list contains summary information for the objects. The bus list contains information about the maximum and minimum voltage values and time. The generator list contains maximum and minimum information about angles and frequency. Note that when the **Process Contingencies** option is set to *One Contingency at a Time*, the list will only apply to the presently active contingency chosen from the **For Contingency** drop-down menu. (See [Transient Stability: Running Multiple Contingencies](#).)

Summary

Time Solution Started

Time that the transient stability analysis started.

Time to Solve (Seconds)

Amount of computing time needed to solve the analysis.

Maximum Angle Difference (Deg)

Maximum angle difference between any two generator rotors at any time during the analysis.

Time of Max. Angle Diff. (Seconds)

Analysis time at which the maximum angle difference occurred.

Newton Solution Results

This contains information about the Newton Solutions needed during the Simulation

Events

The Events tab will show a list of events which occurred during the stability run. The user-defined transient contingency elements will be listed in the event list. In addition, any actions caused by the action of a relay or motor tripping action of one of the stability models may be listed as an event.

Solution Details

This page provides some summary information about the Newton solution results.

Transient Stability Results: Time Values from RAM

The Time Values from RAM are found as a sub-tab under the Results page is found on the [Transient Stability Analysis dialog](#). The table lists values for each of the selected objects and fields as columns and the various time steps as rows. Note that for time-step at which an event (such as a fault) occurs, there may be multiple time steps listed showing the same time. The first listed time step will represent the system state immediately before that time (T-) and the second listed time step will represent the system state immediately after the events occur. There are also options along the left to choose which fields to show as columns as well a column filtering options to use an advanced filter to limit the columns. Finally there is an option to specify whether columns should be sorted first by *Object then Field* or by *Field then Object*. This is all depicted in the following image.

Note that when the **Process Contingencies** option is set to *One Contingency at a Time*, the list will only apply to the presently active contingency chosen from the **For Contingency** drop-down menu. (See [Transient Stability: Running Multiple Contingencies](#).)

Results from RAM

Time Values | Minimum/Maximum Values | Summary | Events | Solution Details

Generator | Bus | Load | Switched Shunt | Branch | DC Transmission Line | Multi-Terminal DC Record | Multi-Terminal DC Converter | Area | Zone | Interface

Column Order: Object then Field

Column Filtering: Filter [Modify...]

☐ Use Area/Zone Filters

Choose Fields to Display

- ☒ Angle
- ☒ Angle, No Shift
- ☒ Frequency
- ☒ Gen Mvar Total
- ☒ Gen MW Total
- ☒ Load Mvar Total
- ☒ Load MW Total
- ☒ V (pu)

Check All | Uncheck All

	Time	Bus Bus1 V (pu)	Bus Bus1 Angle	Bus Bus1 Angle, No Shift	Bus Bus1 Frequency	Bus Bus1 Gen MW Total	Bus Bus1 Gen Mvar Total	Bus Bus1 Load MW Total	Bus Bus1 Load Mvar Total	Bus Bus2 V (pu)	Bus E Ang
119	0.983	1.04	0	0	60	71.6446	27.0385	0	0	1.025	
120	0.992	1.04	0	0	60	71.6446	27.0385	0	0	1.025	
121	1	1.04	0	0	60	71.6446	27.0385	0	0	1.025	
122	1	0.8515	-0.359	-0.359	60	67.9224	285.7476	0	0	0.36	19
123	1.008	0.8505	-0.361	-0.3096	60.0044	66.7124	286.4878	0	0	0.3478	21
124	1.017	0.8496	-0.461	-0.2554	60.0081	65.4383	287.2028	0	0	0.337	24
125	1.025	0.8487	-0.6584	-0.1963	60.0113	64.1064	287.8997	0	0	0.3273	26
126	1.033	0.8478	-0.9554	-0.1347	60.0141	62.7908	288.5995	0	0	0.3184	28
127	1.042	0.847	-1.3429	-0.0624	60.0166	61.2906	289.2665	0	0	0.3111	3
128	1.05	0.8461	-1.8315	0.0097	60.0189	59.8968	289.9628	0	0	0.304	33
129	1.058	0.8453	-2.408	0.094	60.0211	58.3087	290.639	0	0	0.2983	35
130	1.067	0.8445	-3.0847	0.1777	60.0233	56.8585	291.3552	0	0	0.2928	38
131	1.075	0.8437	-3.8466	0.2749	60.0255	55.2052	292.0635	0	0	0.2883	40
132	1.083	0.8429	-4.7072	0.3713	60.0276	53.7211	292.8199	0	0	0.284	43
133	1.092	0.8422	-5.6502	0.4825	60.0299	52.0245	293.5792	0	0	0.2805	45
134	1.1	0.8414	-6.6905	0.5927	60.0321	50.5281	294.3929	0	0	0.2771	47
135	1.108	0.8407	-7.8101	0.7189	60.0345	48.8104	295.2186	0	0	0.2744	50
136	1.117	0.8399	-9.0258	0.844	60.0368	47.3233	296.1033	0	0	0.2717	52
137	1.125	0.8392	-10.3175	0.9864	60.0393	45.606	297.0074	0	0	0.2695	55
138	1.133	0.8384	-11.7035	1.1273	60.0418	44.1501	297.9734	0	0	0.2674	57
139	1.142	0.8377	-13.163	1.287	60.0445	42.4549	298.9647	0	0	0.2656	60
140	1.15	0.8369	-14.7152	1.4448	60.0472	41.0513	300.0193	0	0	0.2639	62
141	1.158	0.8362	-16.3376	1.6226	60.05	39.4003	301.1035	0	0	0.2625	65
142	1.167	0.8354	-18.0517	1.7984	60.0529	38.0697	302.2512	0	0	0.2611	68

Load from Hard Drive File into RAM results specified by Store to RAM Options

Transient Stability Analysis: States/Manual Control

The States/Manual Control page is found on the [Transient Stability Analysis dialog](#). This page provides a means to manually control a transient stability run and view generator and bus states at each step of the process. It will be very helpful in determine which part of the system is causing an instability. It is also helpful to look at initial State Limit Violations as described below. Also note that when you first open the Transient Stability Dialog, Simulator does not immediately initialize the transient stability simulation. This initialization obviously occurs when you start simulation, but it also occurs when you switch to either the States/Manual Control or the [SMIB Eigenvalues](#).

One particular good use of this section is to sort by the *Derivative* column under the All States sub-tab to ensure of that the state derivatives are all near zero at the initialized system state. Note: many induction motor states will never be exactly zero because their initial states must be determined by an iterative process, but in general derivatives should be very near zero. Another good use of this section is to look at the initial condition State Limit Violations.

States/Manual Control

Reset to Start Time

Run Until Specified Time: 5.000 Run Until Time

Do Specified Number of Timestep(s): 1 Number of Timesteps to Do

Transfer Present State to Power Flow

Restore Power Flow Model

Save Time Snapshot

All States | State Limit Violations | Generators | Buses | Transient Stability YBus

	Model Class	Model Type	Object Name	At Limit	State Ignored	State Name	Value	Derivative	Delta X K1
1	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	Angle	-0.5002	0.0000000	0.0000000
2	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	Speed w	0.0000	0.0000000	0.0000000
3	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	Eqp	0.9524	0.0000000	0.0000000
4	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	PsiDp	0.9221	0.0000000	0.0000000
5	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	PsiQpp	0.4132	0.0000000	0.0000000
6	Gen Synch. Machine	GENROU	10261 (REEVE_G1) #1		NO	Edp	0.1807	0.0000000	0.0000000
7	Gen Exciter	TEXS	10261 (REEVE_G1) #1		NO	Feedback	0.0000	0.0000000	0.0000000
8	Gen Exciter	TEXS	10261 (REEVE_G1) #1		NO	Sensed Vt	1.0098	0.0000000	0.0000000
9	Gen Exciter	TEXS	10261 (REEVE_G1) #1		NO	Derivative Controller	0.0000	0.0000000	0.0000000
10	Gen Exciter	TEXS	10261 (REEVE_G1) #1		NO	Integral Controller	1.3239	0.0000000	0.0000000
11	Gen Governor	GGOV1	10261 (REEVE_G1) #1		NO	PElec Measured	0.5000	0.0000000	0.0000000
12	Gen Governor	GGOV1	10261 (REEVE_G1) #1		NO	Governor Differential Control	0.0000	0.0000000	0.0000000
13	Gen Governor	GGOV1	10261 (REEVE_G1) #1		NO	Governor Integral Control	0.5017	0.0000000	0.0000000
14	Gen Governor	GGOV1	10261 (REEVE_G1) #1		NO	Turbine Actuator	0.5017	0.0000000	0.0000000

Reset to Start Time

Click this button to reset the simulation back to the start time as specified on the [Simulation Control](#) page.

Run Until Specified Time

Click this button to run to the specified **Run Until Time**. The Run Until Time should be set in seconds. The simulation is run from the present time to this specified time. The current time is shown in the **Simulation Status** field that is displayed at the top of the [Transient Stability Analysis dialog](#). The Run Until Time must be later than the present simulation time.

Do Specified Number of Timestep(s)

Click this button to run the specified **Number of Timesteps to Do**. The simulation starts at the present time and does the number of timesteps. The current time is shown in the Simulation Status field that is displayed at the top of the [Transient Stability Analysis dialog](#).

Transfer Present State to Power Flow

Click this button to transfer the present state resulting from the transient stability analysis to the power flow model. The present state is not transferred to the power flow model when running the transient stability analysis when not using the manual mode.

Restore Power Flow Model

Click this button to restore the power flow model to the pre-transient stability analysis state. This option can be used when the transient stability state information has been transferred to the power flow model.

Save Time Snapshot

Click this button to save the power flow model to file with the present transient stability results. Transient stability results will only be saved if choosing a file type of either *.pwb or *.aux. If the transient stability state has been transferred to the power flow model, the power flow model is first restored to the pre-transient stability analysis state before saving.

All States

This sub-tab contains a list of every state in the dynamic simulation. The *Value* of each state is listed along with the *Derivative* of the state.

State Limit Violations

This sub-tab contains a list of all the state limit violations in the initialized system.

Generators and Buses

This sub-tab contains a list of all the present terminal values for the generators and buses. These are the same values you can see on the [Transient Stability Data: Object Dialogs](#) and include fields such as Field Voltage, Rotor Angle, Accelerating MW, etc...

In the Generators sub-tab, in addition to getting all the present terminal values for the generators by right-clicking on the Generator entries and choosing Save Two Bus Equivalent from the local menu, you can save the Two Bus system from the selected bus to an infinite bus. When writing out this two bus equivalent model, generators automatically set to AVR = NO In addition when saving the Two Bus equivalent it can save a GENCLS model on the slack bus saved to represent an infinite bus.

Transient Stability YBus

This sub-tab displays the transient stability Y-Bus following the last transient stability analysis timestep.

Transient Stability Analysis: Validation

The Validation page is found on the [Transient Stability Analysis dialog](#).

Because a transient stability simulation is a numerical integration of a set of equations, the numerical integration imposes some restrictions on the input data in order to prevent numerical instability. Numerical instability is instability caused not by the actual power system but by the limitations of the computer algorithms used to perform numerical integration. Simulator provides the ability to both validate the model and auto-correct many of the input values. These features are available on the Validation portion of the Transient Stability Dialog as shown in the following figure. Clicking the **Run Validation** button will run the validation and clicking the **Run AutoCorrection** button will modify many of the input values. In Simulator running the auto correction will permanently change your input data. Running both of these features will generate a list of validation errors, validation warnings, and informational messages. Validation errors are typically related to time constants or machine reactance values as discussed below. Validation errors may also appear for models which have been read in but are not supported by the numerical integration. Validation warnings will appear for things such as a generator model which has no machine model, a generator connected to a bus with zero voltage, or for parameters outside of their expected range. Information messages also appear for parameters outside of the expected range, and when using the Auto Correction tools messages regarding changes to input values will appear.

Element Type	General Type	Model Type	Who Am I	Validation Message
1 Generator	Machine Model	GENROU	WPCC4ST1_ 13.8 (14950) #1	Tqop > 5*Tqopp is recommended.
2 Generator	Machine Model	GENROU	SANTAN 1_ 13.8 (15921) #1	Tqop > 5*Tqopp is recommended.
3 Generator	Machine Model	GENROU	SANTAN 2_ 13.8 (15922) #1	Tqop > 5*Tqopp is recommended.
4 Generator	Machine Model	GENROU	SANTAN 3_ 13.8 (15923) #2	Tqop > 5*Tqopp is recommended.

Unsupported Models

Determine how to proceed if an unsupported transient stability model is defined for any element. An error will prevent the transient stability analysis from running and provide a message. A warning will set the status of any unsupported models to inactive and provide a message. No error or warning will set the status of any unsupported models to inactive but provide no message.

Minimum time constant size as multiple of time step

For limits on time constants as discussed below, this is the multiple of the time step used in the validation routines.

Time of Last Validation

This shows the time that the last validation check was run. This time is either from the last time that the process was run manually or part of the check done at the beginning of a transient stability analysis run.

Errors: and Warnings:

Provides a count of the number of errors and warnings that have been found. Errors will prevent the transient stability analysis from being run.

Summary Tables

This page is divided into three sub-tabs: **Validation Errors**, **Validation Warnings**, and **Informational Messages**. The **Validation Errors** page provides a list of any objects that have errors and provides a description of the error. Any errors will prevent the transient stability analysis from being run. The **Validation Warnings** page also provides a list of any objects that have problems and provides a description of the error. The difference is that warnings are not considered to be severe enough to prevent the transient stability analysis from being run. The **Informational Messages** page lists objects and messages about any errors or warnings that have been auto-corrected by the software.

Common Validation Messages

Minimum time constant size as multiple of time step

The most common limitations are on time constants in the various dynamic models. There are several common restrictions on a time constant. For a simple integration block $[1/(sT)]$, the time constant must be greater than a specified multiple of the integration time step and cannot be zero.

The second common type of restriction on a time constant, such as for a filter block $[1/(1+sT)]$, is similar except that it also allows the time step to be zero. If the time step is zero then the filter block is ignored in the integration and results in an ignored state as discussed in [Transient Stability Numerical Integration](#).

A third common type of restriction applies across two different time constants such as for a lead-lag block $[(1+sT1)/(1+sT2)]$. For a lead-lag block the denominator time constant must be greater than a multiple of the time step. It can also be zero, but if it is zero, then the causality requires that the numerator's time constant also be zero.

Model validation like this must be built into the software throughout to avoid numerical instability; or as in the case of violating causality, we must avoid breaking fundamental laws of physics. For Simulator our testing has shown that for most model time steps must be at least 2-4 times the integration time step when using our 2nd order Runge-Kutta numerical integration technique.

Machine Model Reactance Validation

Another set of data which should be validated is related to the various reactance values of synchronous machine models. These reactance values are specified on the d-axis and q-axis and are referred to as synchronous reactance (X_d and X_q), transient reactance (X_{dp} and X_{qp}), subtransient reactance (X_{dpp} and X_{qpp}), and leakage reactance (X_l). In order for the equations which model the machine to be numerically stable, the reactance must obey the following relationships: $[X_l > X_{qpp} > X_{qp} > X_q]$ and $[X_l > X_{dpp} > X_{dp} > X_d]$. A violation of any of these relationships can cause numerical instability and also just fundamentally does not make sense. Despite this, these types of errors are very common.

When encountering models which do not obey these relationships, PowerWorld Simulator will perform the following error checking, and when using auto-correction will make the following changes to the input data.

if $X_{qp} > X_q$ then $X_{qp} = 0.8 * X_q$

if $X_{dp} > X_d$ then $X_{dp} = 0.8 * X_d$

if $X_{qpp} > X_{qp}$ then $X_{qpp} = 0.8 * X_{qp}$

if $X_{dpp} > X_{dp}$ then $X_{dpp} = 0.8 * X_{dp}$

if $XI > X_{qpp}$ then $XI = 0.8 * X_{qpp}$

if $XI > X_{dpp}$ then $XI = 0.8 * X_{dpp}$

As a reminder though, running the auto-correction in Simulator will permanently change your input data.

Transient Stability Dialog: SMIB Eigenvalues (Single Machine Infinite Bus)

The SMIB Eigenvalues are found on the [Transient Stability Analysis dialog](#).

The Single Machine Infinite Bus (SMIB) Eigenvalue Analysis is another model error checking tool which was added to Simulator. In PowerWorld's own testing, this has proven invaluable while performing model data input checking. The SMIB eigenvalue tool internally builds a dynamic model of one generator connected to an infinite bus. All of that generator's dynamic models (machine model, exciter, governor, and stabilizer) are included in this model. A linear matrix of all the dynamic states is constructed at the steady state solution and eigenvalue and eigenvector analysis is run on this matrix.

These features are available on the SMIB Eigenvalues portion of the Transient Stability Dialog. To generate these results, click on the **Run SMIB Eigen Analysis** button. After running the SMIB Eigen Analysis, it is very useful to sort the results descending by the Max Eigenvalue column to show any positive eigenvalues (as shown in the first figure below), or to sort ascending by Min Eigenvalue to show negative eigenvalues with a large magnitude (as shown in the second figure below). Positive eigenvalues represent potentially unstable system states, while very large negative eigenvalues represent extremely fast system states which can cause numerical instability. Often times the large negative eigenvalues will be caused by particular exciter models such as the EXST1_GE and REXS models which contain extremely fast feedback loops. The particular models require special consideration in the numeric integration algorithm because of this and are discussed more in the [Transient Stability Numerical Integration](#) topic regarding sub-interval integration.

SMIB Eigenvalues

Run SMIB Eigen Analysis Re-Initialize Eigenvalue Analysis Last Run: 10/22/2009 5:07:54 PM

Records ▾ Set ▾ Columns ▾ f(x) ▾ Options ▾

	Number of Bus	Name of Bus	ID	Area Name of Gen	Machine	Exciter	Governor	Stabilizer	Calculated	Number of Eigenvalues	Number of Zero Eigenvalues	Max Eigenvalue	Min Eigenvalue	Eigenvalue
1	30400	MUSKEG_R_13.8_1001	1	AESO	GENROU	ESAC7B	GGOV1	PSS2A	YES	41	10	7.8257	-54.5192	0.0000 + j0.0
2	30398	MUSKEG_R_13.8_1101	1	AESO	GENROU	ESAC7B	GGOV1	PSS2A	YES	41	10	6.6659	-54.3646	0.0000 + j0.0
3	33756	ROCK_CRK_13.8_1001	1	PG&E	GENSAL	EXDC4	HYGOV		YES	11	0	0.7723	-39.7923	-0.0178 + j0.0
4	4845	G_COULEE_13.8_7012	1	BPA	GENSAL	EXST1_GE	G2WSCC	PSS5B	YES	39	20	0.7175	-100.0068	0.0000 + j0.0
5	4846	G_COULEE_13.8_7011	1	BPA	GENSAL	EXST1_GE	G2WSCC	PSS5B	YES	39	20	0.7175	-100.0068	0.0000 + j0.0
6	20637	CUSHMAN_13.2_1001	1	TCP	GENTPF	EXST1_GE	GPWSCC	WSCCST	YES	24	6	0.5038	-515.0048	0.0000 + j0.0
7	16056	NAUGHTON_20_2001	1	UTAH	GENROU	EXST2A	IEEEG1	IEEEEST	YES	24	8	0.4101	-39.5728	0.0000 + j0.0
8	27362	PCANYON_13.8_4002	1	BC-HYD	GENSAL	EXST1_GE	PIDGOV	IEEEEST	YES	24	6	0.3948	-92.1825	0.0000 + j0.0
9	28141	CORRALIN_7.2_4001	1	BC-HYD	GENSAL	EXDC4	GPWSCC		YES	14	1	0.2586	-146.2211	0.0000 + j0.0
10	28143	CORRALIN_7.2_4003	1	BC-HYD	GENSAL	EXDC4	GPWSCC		YES	14	1	0.2585	-146.2211	0.0000 + j0.0
11	6777	LOW_MON_13.8_1006	1	BPA	GENSAL	EXDC1	IEEEG3_GE	PSS2A	YES	33	9	0.2450	-129.9723	0.0000 + j0.0
12	6780	LOW_MON_13.8_1009	1	BPA	GENSAL	EXDC1	IEEEG3_GE	PSS2A	YES	33	9	0.2447	-129.1922	0.0000 + j0.0
13	21684	MAINCANA_13.8_1004	1	AVA	GENTPF	EXST1_GE	GPWSCC		YES	18	3	0.0299	-64.0989	0.0000 + j0.0
14	5336	HOPKIDG_34.5_514	1	BPA	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0224	-12091.8281	0.0000 + j0.0
15	5337	HOPKIDG_34.5_504	1	BPA	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0224	-11964.9180	0.0000 + j0.0
16	20006	WILD_HRS_34.5_511	1	PSE	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0213	-9677.5439	0.0000 + j0.0
17	20008	WILD_HRS_34.5_501	1	PSE	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0213	-9677.5439	0.0000 + j0.0
18	38283	SANTAN_13.8_1023	1	ARIZON	GENTPF	EXAC2	IEEEG1	PSS2A	YES	36	10	0.0129	-59.9967	0.0000 + j0.0
19	27488	REVELSTK_16_1000	1	BC-HYD	GENSAL	EXST1_GE	PIDGOV	IEEEEST	YES	24	8	0.0115	-42.0943	0.0000 + j0.0

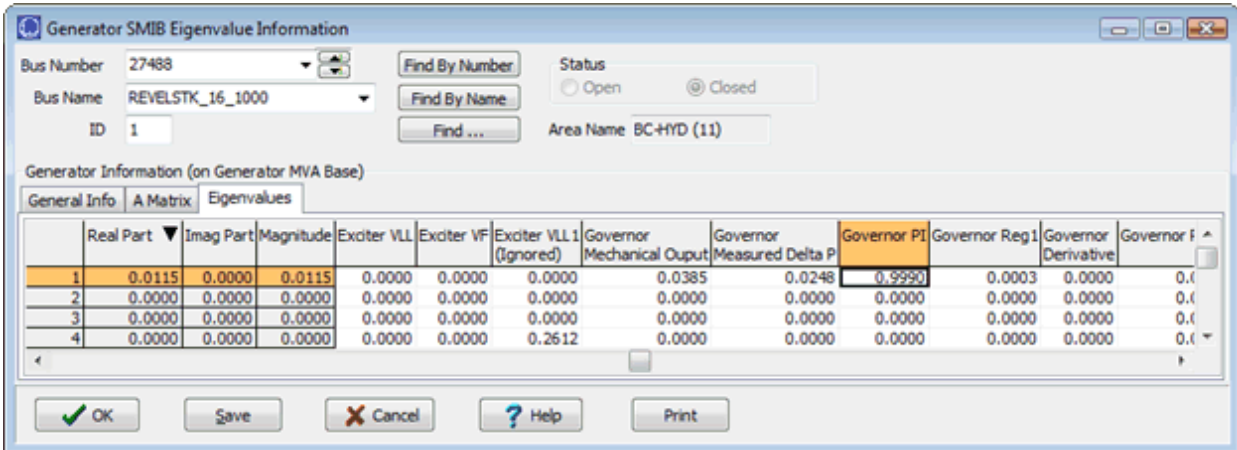
SMIB Eigenvalues

Run SMIB Eigen Analysis Re-Initialize Eigenvalue Analysis Last Run: 10/22/2009 5:07:54 PM

Records ▾ Set ▾ Columns ▾ f(x) ▾ Options ▾

	Number of Bus	Name of Bus	ID	Area Name of Gen	Machine	Exciter	Governor	Stabilizer	Calculated	Number of Eigenvalues	Number of Zero Eigenvalues	Max Eigenvalue	Min Eigenvalue	Eigenvalue
1	5336	HOPKIDG_34.5_514	1	BPA	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0224	-12091.8281	0.0000 + j0.0
2	5337	HOPKIDG_34.5_504	1	BPA	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0224	-11964.9180	0.0000 + j0.0
3	20006	WILD_HRS_34.5_511	1	PSE	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0213	-9677.5439	0.0000 + j0.0
4	20008	WILD_HRS_34.5_501	1	PSE	GENWRI	EXWTG1	WINDTRB		YES	9	1	0.0213	-9677.5439	0.0000 + j0.0
5	18511	ELECTHTS_2.3_1109	1	PSE	GENSAL	EXST1_GE			YES	10	3	-1.3698	-2601.8420	0.0000 + j0.0
6	18510	ELECTHTS_2.3_1110	1	PSE	GENSAL	EXST1_GE			YES	10	3	-1.5434	-2101.6980	0.0000 + j0.0
7	18512	ELECTHTS_2.3_1108	1	PSE	GENSAL	EXST1_GE			YES	10	3	-1.6298	-2101.6653	0.0000 + j0.0
8	8330	PEBUSPRG_34.5_1003	1	BPA	GENWRI	EXWTG1	WINDTRB		YES	9	1	-0.0059	-1639.1625	0.0000 + j0.0
9	33956	SHASTA_13.8_1002	1	PG&E	GENSAL	EXST1_GE	HYG3	PSS5B	YES	39	17	-0.0375	-999.9999	0.0000 + j0.0
10	33958	SHASTA_13.8_1004	1	PG&E	GENSAL	EXST1_GE	HYG3	PSS5B	YES	39	17	-0.0375	-999.9999	0.0000 + j0.0
11	33957	SHASTA_13.8_1003	1	PG&E	GENSAL	EXST1_GE	HYG3	PSS5B	YES	39	17	-0.0375	-999.9999	0.0000 + j0.0
12	29931	KEEPMILL_19_2000	1	AESO	GENROU	EXST1_GE	IEEEG1	IEEEEST	YES	24	8	-0.5061	-982.5344	0.0000 + j0.0
13	29930	KEEPMILL_19_1000	1	AESO	GENROU	EXST1_GE	IEEEG1	IEEEEST	YES	24	8	-0.4019	-900.3795	0.0000 + j0.0
14	21200	BOX_CANY_115_122	1	AVA	GENSAL	REXS	IEEEG3_GE		YES	19	4	-0.0032	-609.8544	0.0000 + j0.0
15	34698	CASTAIC_18_4001	1	LADWIP	GENSAL	EXST1_GE	PIDGOV	IEEEEST	YES	24	7	-0.0183	-600.5581	0.0000 + j0.0
16	34422	USWP_RUS_20_2010	1	PG&E	MOTOR1				YES	5	0	-7.3748	-597.0544	-14.5764 + j0.0
17	34790	HARBOR_13.8_1001	1	LADWIP	GENTPF	EXST1_GE	IEEEG1	IEEEEST	YES	24	9	-0.1000	-535.8034	0.0000 + j0.0
18	15201	BENLOMOND_13.8_13.8	1	UTAH	GENTPF	EXST1_GE			YES	11	3	-0.7241	-515.2556	0.0000 + j0.0
19	15682	GOSHEN_13.8_1000	1	UTAH	GENTPF	EXST1_GE			YES	11	3	-0.4574	-515.2236	0.0000 + j0.0

In addition to getting the eigenvalue information by right-clicking on the SMIB entries and choosing Show SMIB dialog from the local menu, you can open a dialog which shows you the A matrix and the details regarding the eigenvalues and participation factors determined from the eigenvector analysis. The participation factors indicate which particular system states are most contributing to the particular eigenvalue. The following shows an example of the Eigenvalues for one of the generators which had a positive eigenvalue.



In some testing this small positive eigenvalue always appeared for these hydro power plants. When looking at the eigenvector analysis though it pointed us toward looking at the PI feedback loop in the PIDGOV governor model. In looking at the input data, it was seen that the Rperm (droop) of those governors had a negative value which ultimately leads to an unstable governor. In this manner it validated the SMIB tool as correctly showing the input data which led to the positive eigenvalue.

Also by right-clicking on the SMIB entries and choosing Save Two Bus Equivalent from the local menu, you can save the Two Bus system from the selected bus to an infinite bus. When writing out this two bus equivalent model, generators automatically set to AVR = NO In addition when saving the Two Bus equivalent it can save a GENCLS model on the slack bus saved to represent an infinite bus.

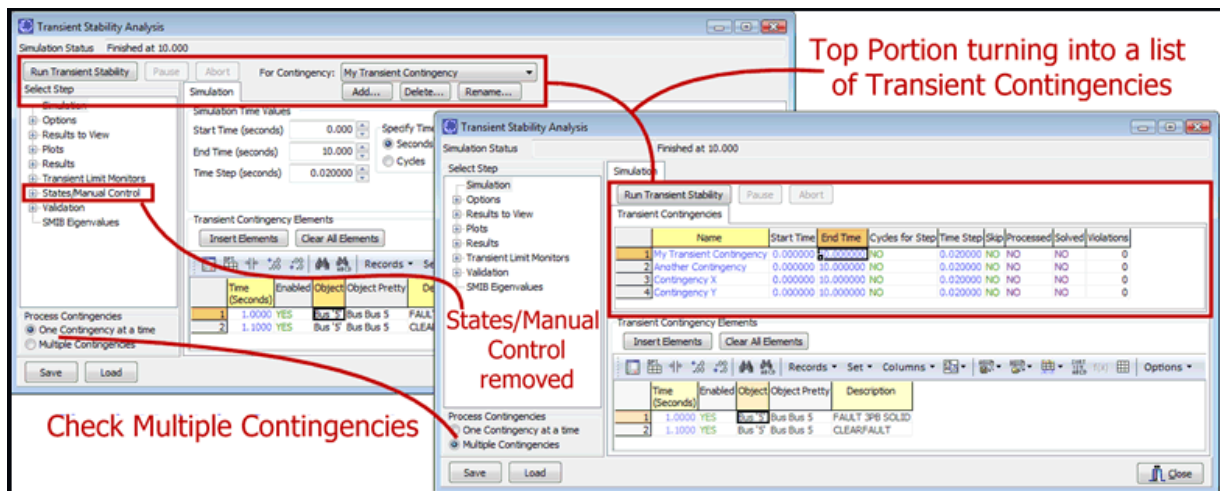
Transient Stability: Running Multiple Contingencies

Most of the help documentation discusses how [Transient Stability Dialog](#) functions when you are simulating only a single transient contingency event. When changing the **Process Contingencies** option to *Multiple Contingencies* instead, the dialog will change in small ways throughout. This topic discusses how the dialog behaves differently when running multiple contingencies.

The ultimate goal of the transient stability tool is to run multiple potential transient contingencies to determine if any violations of any standards has occurred. When running multiple contingencies, our design goal was to make a transient stability run very similar to running a contingency analysis run using repeated power flow solutions. The transient stability interface has been designed to provide the ability to simulate and interact with multiple stability runs simultaneously. Obviously the speed of such a simulation will be much slower than for contingency analysis, but the goal for the tool should be to emulate the contingency analysis environment. These sections discuss three topics related to achieving this goal: user interface implications, data storage, and the use of transient limit monitors.

General User Interface Changes

When choosing to process multiple contingencies, the [Simulation](#) portion of the Transient Stability dialog into a form which looks very similar to PowerWorld's existing contingency analysis tool. At the top will be a list of the transient contingency definitions with columns for the Start Time, End Time, Cycles for Step, and Time Step. When you select a particular transient contingency from this list, the list at the bottom of the dialog will show the particular elements which are part of the selected transient contingency. This is shown in the figure below.



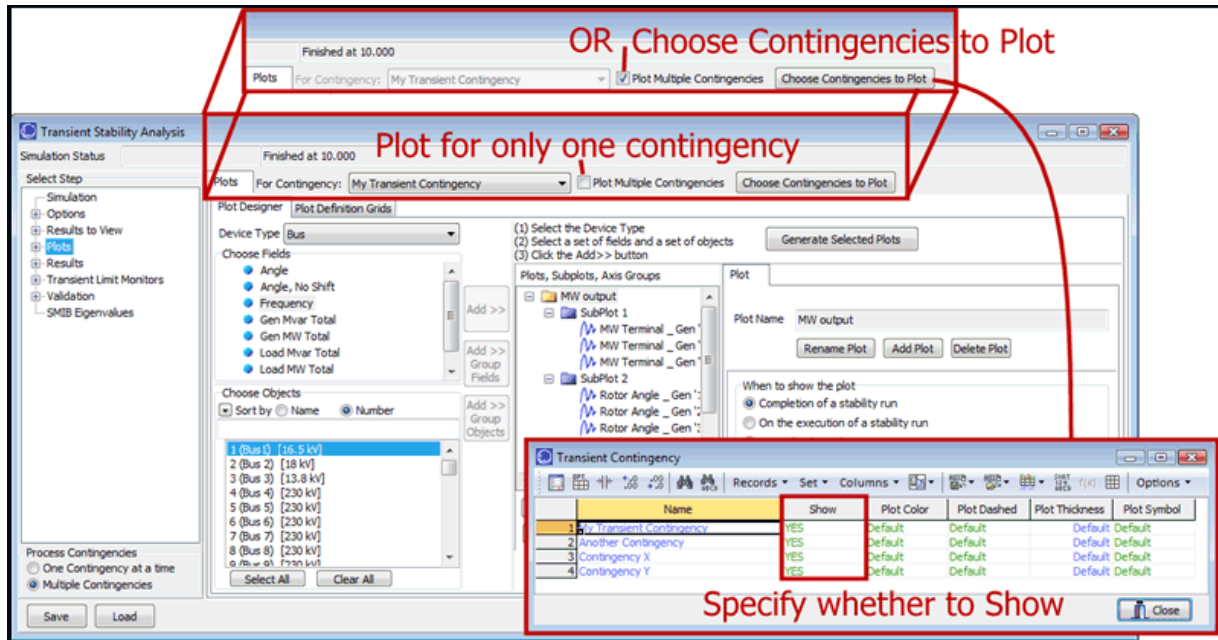
Also note that the [States/Manual Control](#) portion of the dialog is removed. The assumption is that you will not want to manually step through a stability run a few time steps at a time if you're running multiple transient contingencies. The [Plots](#) and [Results](#) (for viewing tabular result) are changed when interpreting multiple contingency analysis runs. These will be discussed shortly.

Other portions of the dialog have very few changes. The [Options](#) do not change at all and will apply to all transient contingencies. The [Results Storage](#) will also apply to all transient contingencies. [Validation](#) and [SMIB Eigenvalues](#) only apply to the initial steady state system so they will also not change. [Transient Limit Monitors](#) do not change because they apply to all contingencies and each of the Transient Limit Violations always reference a particular transient contingency.

Plots for Multiple Contingency Results

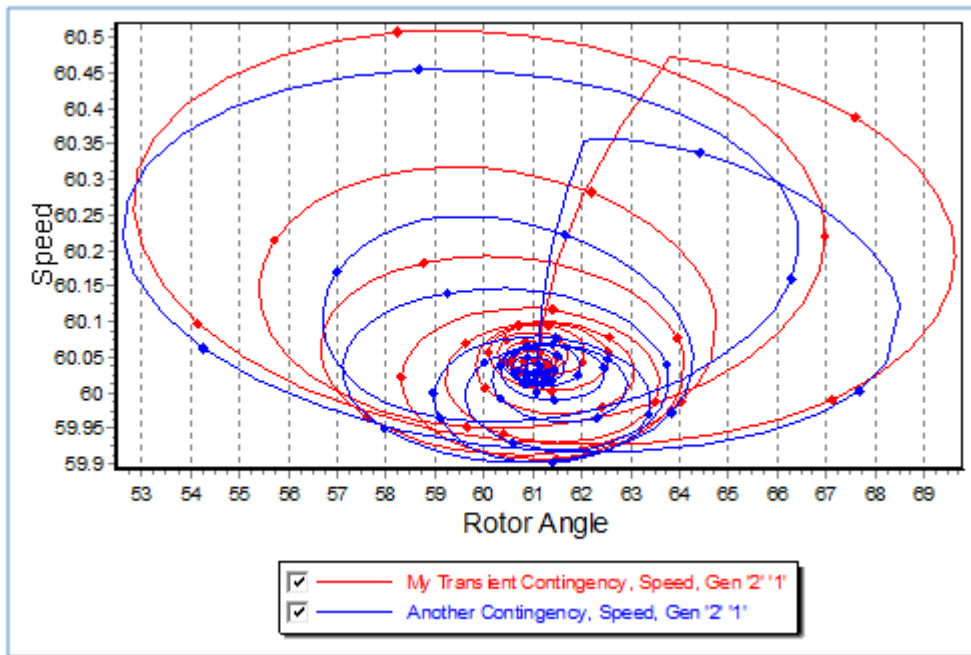
The Plots portion of the Transient Stability dialog changes slightly when processing multiple contingency results. By default, a dropdown allows you to choose for which transient contingency a

plot will be drawn. When creating a plot for only one contingency the behavior is identical as described in Section 6. You may also check the box **Plot Multiple Contingencies** which will modify the behavior of plotting. When doing this you should then click the button **Choose Contingencies to Plot** to **Plot** which brings up a list of all the Transient Contingencies defined in the case. This is all depicted in the following figure.



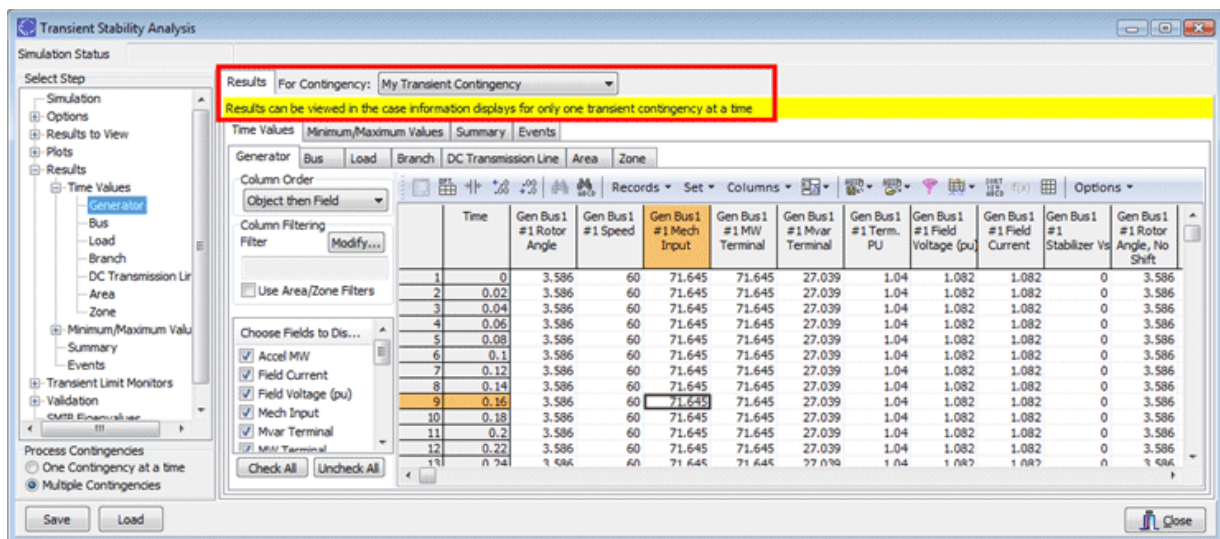
On the list of contingencies there will be five new columns with values that may be specified and whose affect is described as follows. Setting **Show** to YES will cause plot series for that contingency to be generated. If a plot has 10 subplot series inside of it and you choose to show results for 4 transient contingencies, then the resulting chart will contain each of those 10 plot series for each of the 4 contingencies and thus contain 40 plot series. Changing the **Plot Color** will allow each plot series for that particular contingency to use this color, overriding what was specified with the plot definition. Similarly the **Dashed**, **Thickness**, and **Point Symbol** may be changed with the resulting attribute applied to all plot series for the particular contingency overriding what is specified with the plot definition.

In this way you can generate a plot showing multiple traces from multiple transient contingency simulations. An example is shown below of a state space plot for two different faults, with a different color used for each contingency.



Tabular Results for Multiple Contingency Analysis Simulations

Showing of tabular results was already extremely intricate for looking at only one transient contingency event. At this time the user interface has not been designed to support viewing tabular results for multiple transient contingency simulations simultaneously. As a result when going to the Results portion of the transient stability dialog it will be modified slightly to provide a dropdown that allows you to choose for which transient contingency to view results. This is shown in the figure below. From this point the behavior is identical as described in [Transient Stability Results: Time Values](#).



Distributed Computing Add-Ons

The Distributed tools are available as an add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for more details. The use of Distributed tools in Simulator requires the use of the SimAuto add-on as well

Some tools in Simulator have been enabled to use Simulator's Distributed Computing capabilities. In order to use Distributed Computing a list of computers is provided to Simulator which are available for distributed computing. This list is managed in one place in Simulator and applies to all tools in Simulator that can be enabled for distributed computing. Tools which can use Distributed Computing are as follows:

- [Contingency Analysis](#) : can distribute groups of contingencies
- [Available Transfer Capability](#) : can distribute multiple-scenario ATC calculations
- [Transient Stability](#) : can distribute transient contingencies
- QV Curves : planned for later release but not completed yet

Along with this list of computers, appropriate login authentication must be provided for each remote computer to ensure that the Windows operating system security allows the remote processes to be started and results to be passed back to Simulator. These passwords are then encrypted and decrypted by Simulator using a Master Password as described below. For more technical details about password security please contact PowerWorld.

Buttons commonly available for distributed computing are described as follows.

Insert Computer

Inserts a new computer into the list of computers available for distributed computing.

Verify Computers Available

Checks each computer in the list of distributed computing machines for availability, and sets the Enabled, Available, and Cores fields accordingly.

Enter Master Password

Allows the user to enter the password used to decrypt all of the distributed machine login credentials. If the master password has not been defined, then the user is asked to provide a new master password.

Forget Master Password

Tells Simulator to forget the already entered master password. The user will not be able to perform a distributed analysis until the master password has been re-entered.

Change Master Password

Tells Simulator to re-encrypt all credentials using a new master password.

Reset all authentication

Clears all authentication information including the master password and all distributed machine login credentials.

To add login authentication information for a particular Distributed Computer, go to the **Records Menu** or the right-click local menu under **Distributed Computer records** and choose the following options.

Add or change authentication info...

Adds or changes login credentials for all currently selected distributed machines. If a master password has not been defined, the user will be prompted to provide a new master password.

If the master password has been defined but not entered, the user will be prompted to enter the current master password.

Note: Adding authentication for the local machine is not needed and should not be provided. If authentication information is added for the local machine it will result in an "Access is denied" error from Windows.

Remove authentication info

Removes login credentials for all currently selected distributed machines.

The columns with the Distributed Computer List are as follows

Computer Name

Name of the computer on the local network.

Auth Info Stored?

Column will say YES if login authentication information is stored with this computer.

Processes

Specifies the number of processes which will be started on this when performing distributed computer. For computers with multiple cores and/or multiple processors it may be advantageous to execute multiple processes.

Enabled

Set this to NO to disable the use of this computer when using distributed computing. Set to YES to use the computer.

Available

When the **Verify Computers Available** button is clicked, Simulator will determine this field automatically. It shows whether the remote computer is available for use in distributed computing.

Cores

Shows the number cores available on the computer.

Errors

Shows the number of errors encountered when trying to use the computer for distributed computing.

Max # Errors


Enter a number for the maximum number of errors which can be encountered with a computer before the distributed computing will stop attempting to use the computer for computation. By default this value is set to 1 because it is assumed that a failure will

Domain, Password, Username


Shows the domain, username, and password of the login information in an encrypted form. These values can be saved and loaded from an AUX file in encrypted form but you must not attempt to edit them directly. They must be encrypted and decrypted through the Master Password at all times, thus to make changes to these you must use the **Add or change authentication info** and **Remove authentication** info local menu buttons only. If authentication fails then Simulator will not use the computer for distributed computing.

This procedure describes how to create a simple power system model using PowerWorld Simulator. This procedure was developed for use with version 13 and later of the package. If you have an earlier version, please contact PowerWorld Corporation at info@powerworld.com for information on upgrading, or visit the website at <http://www.powerworld.com>.



To begin, double-click on the PowerWorld Simulator icon.  This starts Simulator. Simulator is used to create new cases, modify existing cases, and (of course) simulate power systems. In this example, we will build a new case from scratch. To create a case from an existing power flow case instead, please see [Creating a Case from an Existing Power Flow File](#).

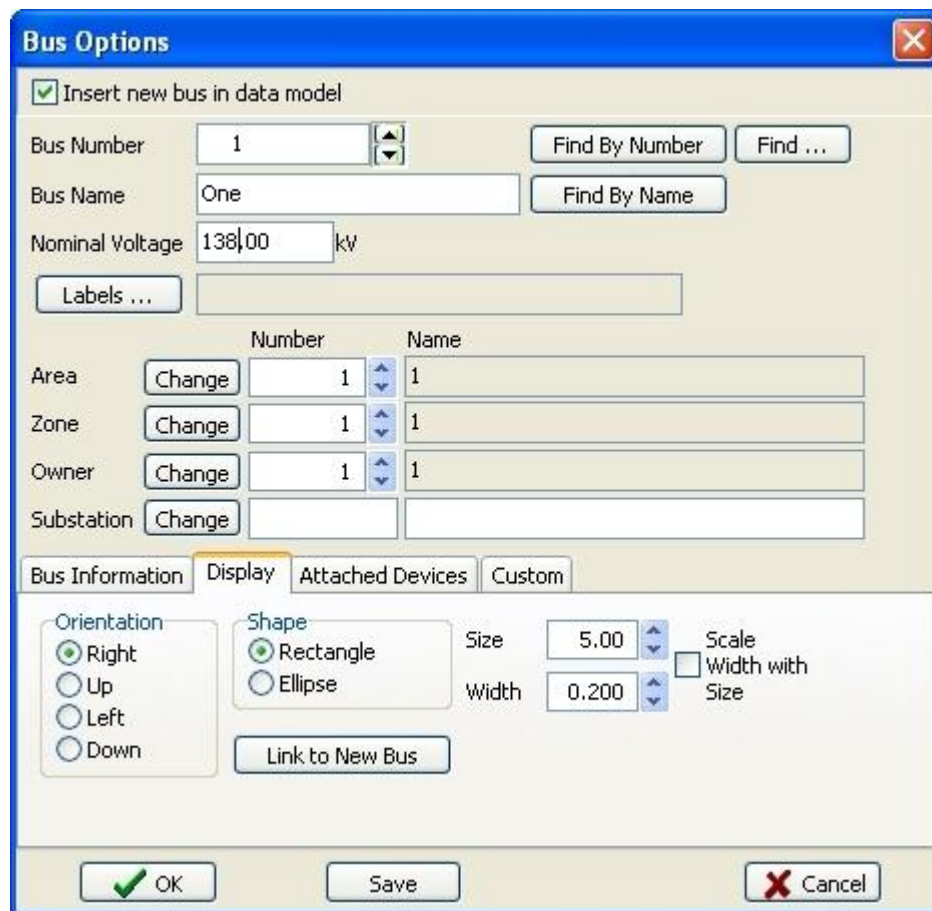


To create a new case, select **New Case** from PowerWorld icon  in the upper left corner of the program. The screen background will turn white, the default background color for new PowerWorld oneline diagram. Oneline diagrams are used in power system analysis to represent the actual three-phase power system using a single line to represent each three-phase device.

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices are connected together. In building a power system model using Simulator, you will draw buses onto the oneline diagram, attach devices such as generators and loads to the buses, and connect different buses together with transmission lines and transformers.

To insert a bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the oneline background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#) (pictured below), which is used to specify the name, orientation, shape, size, width, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to the bus .



The **Bus Options** dialog box is used to configure a new bus. It features a blue title bar with a close button. The main area is divided into several sections:

- General Settings:** Includes a checked checkbox for "Insert new bus in data model". Fields for "Bus Number" (1), "Bus Name" (One), and "Nominal Voltage" (138.00 kV) are present, along with "Find By Number", "Find By Name", and "Find ..." buttons. A "Labels ..." button is also available.
- Location and Identification:** A table with columns "Number" and "Name" for "Area", "Zone", "Owner", and "Substation". Each row has a "Change" button and a dropdown menu.
- Display Tab:** Contains "Orientation" (Right, Up, Left, Down) and "Shape" (Rectangle, Ellipse) radio buttons. "Size" (5.00) and "Width" (0.200) are adjustable. A "Link to New Bus" button is at the bottom.
- Buttons:** "OK", "Save", and "Cancel" buttons are at the bottom.

Bus Options

☒ Insert new bus in data model

Bus Number: 1 Find By Number Find ...

Bus Name: One Find By Name

Nominal Voltage: 138.00 kV

Labels ...

	Number	Name
Area	1	1
Zone	1	1
Owner	1	1
Substation		

Bus Information Display Attached Devices Custom

Bus Voltage

Voltage (p.u.): 1.00 ☒ System Slack Bus

Angle (degrees): 0.00

OK Save Cancel

- The *Bus Number* field automatically displays '1'. Simulator requires that each bus have a unique number. For convenience, accept the default value. Historically, the *Bus Name* field was limited to eight characters. That limitation is no longer imposed in Simulator. Many users still choose to limit the bus name to eight characters by convention and for ease in converting cases to other formats (such as .epc or .raw). For this example, enter '**One**'.
- Next, check the *System Slack Bus* field, which is located in the Bus Voltage portion of the Bus Information Tab. The Slack Bus is a modeling construct that ensures that the power system has enough generation to meet the load. In other words, the slack bus "picks up the slack" caused by system losses or unbalanced generation and load.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. After the dialog box closes, the new bus appears on the oneline at the location you specified.

Tutorial: Inserting a Generator Page 3 of 13

Next we are going to attach a generator to the bus. Generators may be inserted in a manner similar to inserting a bus:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click the bus on the oneline diagram to which you want to attach the generator (for this example, click on the slack bus – bus *One*.) The [Generator Option Dialog](#) (pictured below) will automatically open. The dialog is used to specify the new generator's unit identifier, display size, orientation, MW output and limits, reactive power limits, set point voltage, and cost model.

Generator Options

Bus Number: 1 Find By Number
Bus Name: One Find By Name
ID: 1 Find ...
Area Name: 1 Fuel Type: Unknown
Labels ... Unit Type: UN (Unknown)

Display Information Power and Voltage Control Costs Fault Parameters Owner, Area, Zone, Sub Custom

Power Control
MW Output: 413 Available for AGC Part. Factor: 10.00
Min. MW Output: 0.000 Enforce MW Limits MW Ramp Limit: 20.0
Max. MW Output: 1000.000

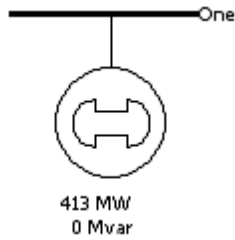
Voltage Control
Mvar Output: 0.000 Regulated Bus Number: 1
Min Mvars: -9900.000 Available for AVR SetPoint Voltage: 1.0000
Max Mvars: 9900.000 Use Capability Curve Remote Reg %: 100.0


MW	Min Mvar	Max Mvar

OK Save Cancel Help

- Every generator must have a *MW Output* specified when the generator is inserted. Make sure the *MW and Voltage Control* tab is selected. Enter '**413**' in the *MW Output* Field. Note: the *MW Output* specified for a generator connected to the system slack bus is arbitrary because the generator's true output depends on system load and losses.
- Select the *Display Information* tab. The *Orientation* field is used to specify the direction the generator will extend from the bus. The *Anchored* checkbox forces the generator to move with its specified bus when repositioning the bus on the oneline.

- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator will appear on the oneline attached to the previously selected bus. The oneline diagram should resemble the image shown below.



To save the work that we have done so far, select **Save Case** from the [Application Button](#), or click on the *Save Case*  button. Before the case is saved, Simulator validates the case to make sure that it does not contain any errors. Results from this validation are displayed in the Message Log display, usually shown in the lower right-hand corner of the display. If the log is not visible, click the **Log** on the Log group under **Tools**. Since we have not yet named the case, the **Save As** dialog is displayed. Enter a file name and select **OK**. By default the case is saved using the [PowerWorld Binary format](#) (*.pwb). When saving the case in the future, you will not have to reenter its name. Simulator also asks you to supply a name for saving the oneline diagram we have been drawing. The oneline diagram files have a default extension of *.pwd, which identifies them as PowerWorld Display files. Supply the same name as you gave to the case. Note that, because the case and the oneline are stored in separate files, multiple onelines can be assigned to the same case, and the same oneline can be used by many cases.

To enter the second bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on the oneline diagram somewhere to the right of the first bus. In the Bus Options Dialog (pictured below) leave the bus number at the default value of **2**, and enter the name '**Two**' in the *Bus Name* field.
- We will model a 200 MW, 100 Mvar load at the bus. Select the *Attached Devices* tab. Under the *Load Summary Information* heading enter '**200**' in the *Base MW* field and '**100**' in the *Base Mvar* field.
- Click **OK** to accept all other default values, close the Bus Options Dialog, and insert the bus.

Bus Options

Bus Number: 2 Find By Number Find ...

Bus Name: Two Find By Name

Nominal Voltage: 138.0 kV

Labels ... no labels

	Number	Name
Area	1	1
Zone	1	1
Owner	1	1
Substation		

Bus Information Display **Attached Devices** Custom Fault Parameters

Load Summary Information

Base MW: 200.0

Base Mvar: 100.0

Add or Edit Bus Load

Shunt Admittance

G (MW): 0.0

B (Mvar): 0.0

OK Save Cancel

At this point, the oneline diagram does not show the load at bus 2, even though it is represented in the power system model (you can confirm this by right-clicking on bus 2, selecting *Bus Information Dialog* from the resulting local menu, and inspecting the *Load Summary Information* fields again).

To draw the load on the oneline diagram:

- Select **Network > Load** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click in the center of this bus. The Load Options Dialog box (pictured below) automatically opens. The *Constant Power MW* and *Mvar* fields confirm that the load is 200 MW and 100 Mvar. In addition to constant power loads, Simulator also allows the modeling of voltage dependent loads.

Load Options

Bus Number: Find By Number

Bus Name: Find By Name

ID: Find ...

Labels ...

Status: ☐ Open ☒ Closed

Area: Number: Name:

Zone: Number: Name:

Substation:

Owner: Number: Name:

☒ Same Owner as Terminal Bus

Load Information:

	Constant Power	Constant Current	Constant Impedance
MW Value	<input type="text" value="200.000"/>	<input type="text" value="0.000"/>	<input type="text" value="0.000"/>
Mvar Value	<input type="text" value="100.000"/>	<input type="text" value="0.000"/>	<input type="text" value="0.000"/>

Display Information:

Display Size:

☒ Scale Width with Size

Display Width:

Pixel Thickness:

Orientation: ☐ Right ☐ Left ☒ Up ☐ Down

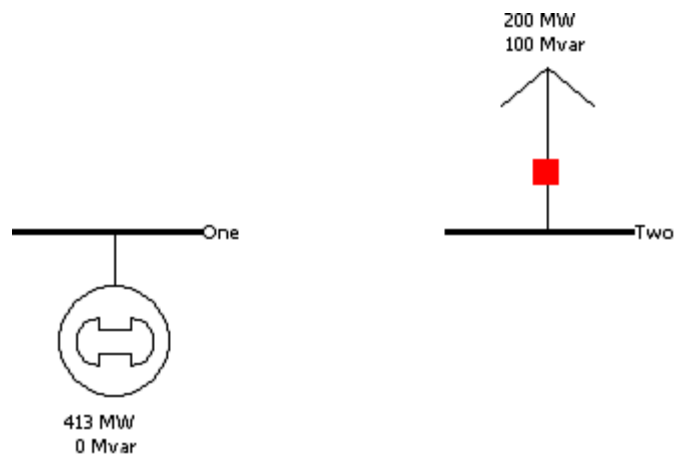
☒ Anchored

- Select **Up** in the *Orientation* field under the *Load Information* tab to make the load point up. Verify that the anchored box is checked to force the load to move with the selected bus.
- Click **OK** to accept the default values for all remaining fields, close the Load Options dialog, and insert the load. A circuit breaker symbol is automatically included with each load.

To move objects on the oneline:

- Left-click on the desired object. Drag and drop the object to the new location by holding the left mouse button down while moving the mouse. Note: you can also move all objects on the oneline simultaneously by left-clicking on the diagram (not on a specific object) then dragging and dropping in the desired location.
- To move bus 2, left click on bus 2 (not on the attached load). Drag the bus to a new location. Note that the load moves with the bus because it is anchored. You can change the location of attached devices connected to a bus, such as generators and loads, by the same procedure.

The oneline diagram should now resemble the image shown below.



Transmission lines are used to connect buses together. To insert a transmission line:

- Select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click at the point where you want the new line to originate. This point is usually located on one of the proposed line's terminal buses. For this example, originate the line at bus One.
- Transmission lines and transformers are drawn as a series of line segments. Without holding down the mouse button, drag the mouse up. Notice that a line segment connected to the point of origin will follow your mouse movements. To terminate a line segment, click the left mouse button. Each time you click the mouse to terminate a line segment, a new vertex is defined for the line. To draw the next line segment, move the mouse to the desired location of the next vertex. Note: the vertices may later be moved or deleted to reshape the line. To create curved lines, hold the left mouse button down while dragging.
- To terminate the final line segment and conclude drawing the line, double click the left mouse button at the desired termination point (bus Two for this example). The termination point is usually the transmission line's other terminal bus.
- The [Branch Options dialog](#) automatically appears (shown below). The dialog should already contain a **1** in the *From Bus Number* field and a **2** in the *To Bus Number* Field. If not, you probably did not have the cursor directly on the bus when you were drawing the line. If this is the case, simply enter the correct bus numbers in the corresponding fields.

Branch Options

Line: From Bus 1 To Bus 2 Circuit 1

Name: One Two

Area Name: 1 (1) 1 (1)

Nominal kV: 138.0 138.0

Labels ... no labels

Find By Numbers Find By Names Find ...

☒ From End Metered

☒ Default Owner (Same as From Bus)

Display Parameters Fault Info Owner, Area, Zone, Sub Custom Stability

Status: ☐ Open ☒ Closed

Length (mi): 0.00

Calculate Impedances >

Per Unit Impedance Parameters

Series Resistance (R): 0.02000

Series Reactance (X): 0.08000

Shunt Charging (B): 0.10000

Shunt Conductance (G): 0.00000

☐ Has Line Shunts Line Shunts

MVA Limits

Limit A	1000
Limit B	0.000
Limit C	0.000
Limit D	0.000
Limit E	0.000
Limit F	0.000
Limit G	0.000
Limit H	0.000

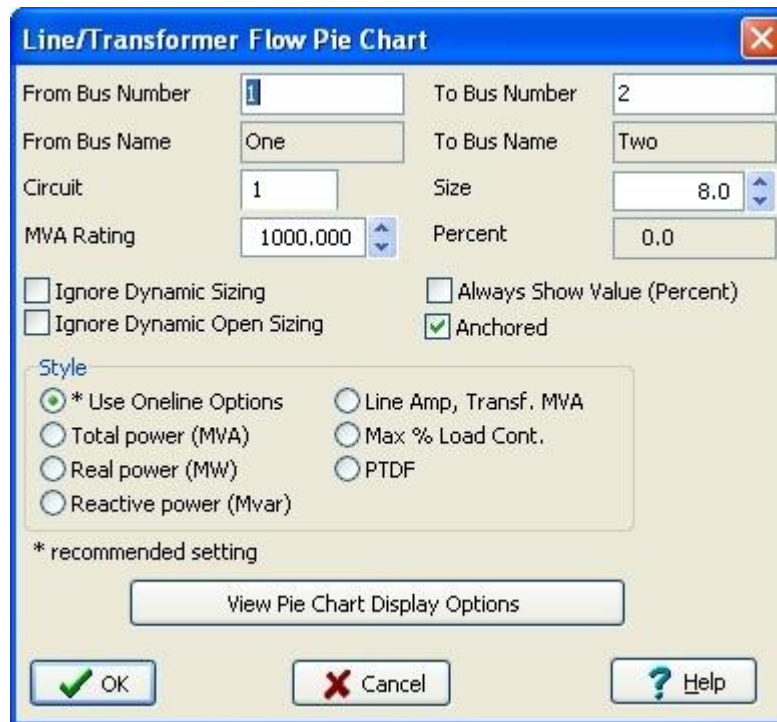
Convert Line to Transformer

OK Save Cancel Help

- The *Series Resistance*, *Series Reactance*, and *Shunt Charging* fields are used to enter the per unit parameters associated with the line. The *Shunt Charging* field contains the total per unit charging capacitance for the line. Enter **0.02** in the *Resistance* field, **0.08** in the *Reactance* field and **0.1** in the *Shunt Charging* field.
- The *Limit (MVA)* fields contain the MVA ratings for the line; enter a value of 1000 in the *Limit A (MVA)* field.
- Click **OK** to accept all remaining field default values, close the Transmission Line/Transformer Dialog, and insert the new line.

By default, the transmission line is anchored to both terminal buses. If you try to move bus 2, the transmission line should move with it.

When the line is drawn it automatically it has a line flow pie chart included. You can include additional line flow pie charts by selecting **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then clicking near a line. The Line/Transformer Flow Pie Chart dialog box appears (shown below). Make sure that the From Bus and To Bus have the correct numbers, that the MVA rating is correct, and that Anchored is checked. You may change the size of the pie chart by typing in a value or using the arrows.



The dialog box is titled "Line/Transformer Flow Pie Chart" and contains the following fields and options:

From Bus Number	1	To Bus Number	2
From Bus Name	One	To Bus Name	Two
Circuit	1	Size	8.0
MVA Rating	1000.000	Percent	0.0

Options:

- ☐ Ignore Dynamic Sizing
- ☐ Ignore Dynamic Open Sizing
- ☐ Always Show Value (Percent)
- ☒ Anchored

Style section:

- ☒ * Use Online Options
- ☐ Total power (MVA)
- ☐ Real power (MW)
- ☐ Reactive power (Mvar)
- ☐ Line Amp, Transf. MVA
- ☐ Max % Load Cont.
- ☐ PTDF

* recommended setting

View Pie Chart Display Options

Buttons: OK, Cancel, Help

Color and behavior of pie charts are set in the *Online Display Options* display; right-click anywhere in the background of the online, select *Online Display Options* from the pop-up menu, then select the *Pie Charts* tab (pictured below).

Online Display Options

Select Option Category

- Animated Flows
- Display Object Options
- Display Options
- Geography/Coordinates
- Grid/Highlight Unlinked
- Memo
- Pie Charts/Gauges**
- Substations
- ThumbNail View

Pie Charts/Gauges

☒ Show Pie Charts/Gauges in Run Mode

☒ Only Show Pie Charts/Gauges if relevant data exists

Lines Interfaces **Pie Chart/Gauge Styles** General Options (All)

Pie Chart Style

☐ Total power (MVA)
 ☒ Amp, Transf. MVA
 ☐ Real power (MW)
 ☐ Max % Load Cont.
 ☐ Reactive pwr (Mvar)
 ☐ PTDF

☐ Always Use Limiting Flow

Gauge Style

☐ Total power (MVA)
 ☒ Amp, Transf. MVA
 ☐ Real power (MW)
 ☐ Max % Load Cont.
 ☐ Reactive pwr (Mvar)
 ☐ PTDF

☐ Display Gauge Values in Percent

Color, Size, and Percentage

MVA MW Mvar Amp CTG PTDF Open Parameters

Show Value Percent 80.0

Normal Size Scalar 1.0

Normal Color

☒ Use Discrete Map

Warning/Limit Scalars and Colors

Percent	Scalar	Color
80.00	1.50	
100.00	2.00	

☐ Make normal color the same as the line to which the pie chart or gauge is anchored (if any)
 ☐ Only Apply Warning/Limit Colors And Resizing to Monitored Elements

☒ OK
 Apply
 Save Options to Case...
 ☒ Cancel
 ? Help

Circuit breakers are used to control the status of the line. (If the line already has circuit breakers at each end, then Simulator has been instructed to insert circuit breakers automatically. You can configure this option from the [Default Drawing Options Dialog](#)).

To insert a Circuit Breaker:

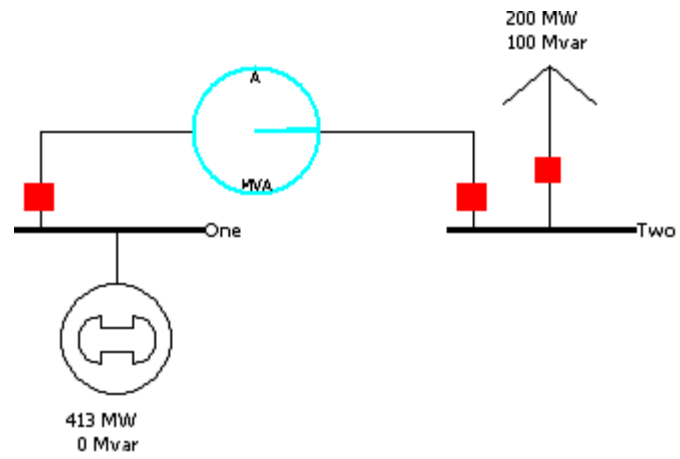
- Click somewhere on the line near bus One then select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, then click on the line near bus One. You should immediately see the Circuit Breaker Options dialog (shown below) with the From Bus and To Bus fields correctly set to '1' and '2'. If they are '0', enter the correct value. Set the Size field to '1' (you can either enter a 1, or use the spin arrows to change the value).
- Click **OK** to insert the circuit breaker.



The image shows a dialog box titled "Circuit Breaker Options". It contains several fields and groups of radio buttons. The "Near Bus" field is set to "1", the "Far Bus" field is set to "2", and the "Circuit" field is set to "1". The "Size" field is a spinner set to "1.00". The "Orientation" group has radio buttons for "Right", "Up", "Left", and "Down", with "Down" selected. The "Anchored" checkbox is checked. The "Shape" group has radio buttons for "Use Default", "Rectangle", "Ellipse", "Triangle", "Rounded Rectangle", "Diamond", and "Switchgear", with "Rectangle" selected. The "Status" group has radio buttons for "Open" and "Closed", with "Closed" selected. The "Pixel Thickness" field is a spinner set to "1". At the bottom are "OK" and "Cancel" buttons.

In Simulator, the location of the circuit breaker does not matter, because changing the status of the circuit breaker changes the status of the entire line. However, since most transmission lines have circuit breakers at each end, we will also place a circuit breaker near bus 2. To accomplish this, repeat the above process near bus 2.

Save your case. Your oneline should now look similar to the image below.



To insert a transformer we first need to insert a bus at a different voltage level. Insert a new bus in the bottom of the oneline, named **Three**, and enter **69** KV for its nominal voltage in the Bus Options dialog box.

To insert a transformer between buses 2 and 3:

- Either go to **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on bus 2, and then draw a line to bus 3 as you did for a transmission line. The [Branch Options dialog](#) automatically appears.

Branch Options

Transformer Number: 2, Name: Two, Area Name: 1 (1), Nominal kV: 138.0

From Bus: 2, To Bus: 3, Circuit: 1, Name: Three

Find By Numbers, Find By Names, Find ...

☒ From End Metered

☒ Default Owner (Same as From Bus)

Labels ... no labels

Display Parameters Transformer Control Fault Info Owner, Area, Zone, Sub Custom Stability

Status: ☐ Open, ☒ Closed

Length (mi): 0.00

Calculate Impedances >

Per Unit Impedance Parameters

Series Resistance (R)	0.02000
Series Reactance (X)	0.08000
Shunt Charging (B)	0.10000
Shunt Conductance (G)	0.00000
Magnetizing Conductance	0.000000
Magnetizing Susceptance	0.000000

☐ Has Line Shunts, Line Shunts

MVA Limits

Limit A	1000.000
Limit B	0.000
Limit C	0.000
Limit D	0.000
Limit E	0.000
Limit F	0.000
Limit G	0.000
Limit H	0.000

Convert Transformer to Line

OK Save Cancel Help

- On the **Parameters** tab, enter **0.02** for *Series Resistance*, **0.08** for *Series Reactance*, **0.1** for *Shunt Charging* and **1000** for *Limit A (MVA)*.
- Select the **Transformer Control** tab. Note that the Off-nominal Turns Ratio displays **1.000**. The true transformer turns ratio does not need to be specified as it is automatically determined by the

ratio of nominal voltages between the *From Bus* and *To Bus*. The *Off-nominal Turns Ratio* is used to adjust the transformer tap setting on per-unit values of bus voltages as referenced to their respective base values for per-unit calculation.

- Click **OK** to accept the default values, close the dialog and insert the transformer.
- Repeat this procedure to add a transformer between buses 1 and 3.

Add a 400 MW, 200 Mvar load to bus 3:

- Right-click on bus 3 and select **Bus Information Dialog**. Select the **Attached Devices** tab. Click **Add or Edit Bus Load**.
- Under *Constant Power* enter **400** in the *MW Value* field. Enter **200** in *Mvar Value* field. Click **OK**.
- Note that *Base MW* and *Base Mvar* display the respective values. Click **OK**.

The load is now attached to the bus even though it is not displayed on the oneline. To display the load as an object on the oneline, you can either use the **Auto Insert** feature or follow the procedure utilized earlier. To **Auto Insert** the load:

- Select **Auto Insert > Loads...** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. The *Automatic Insertion of Loads* dialog opens automatically.
- Click **OK** to accept the default values and insert the load object on the oneline.

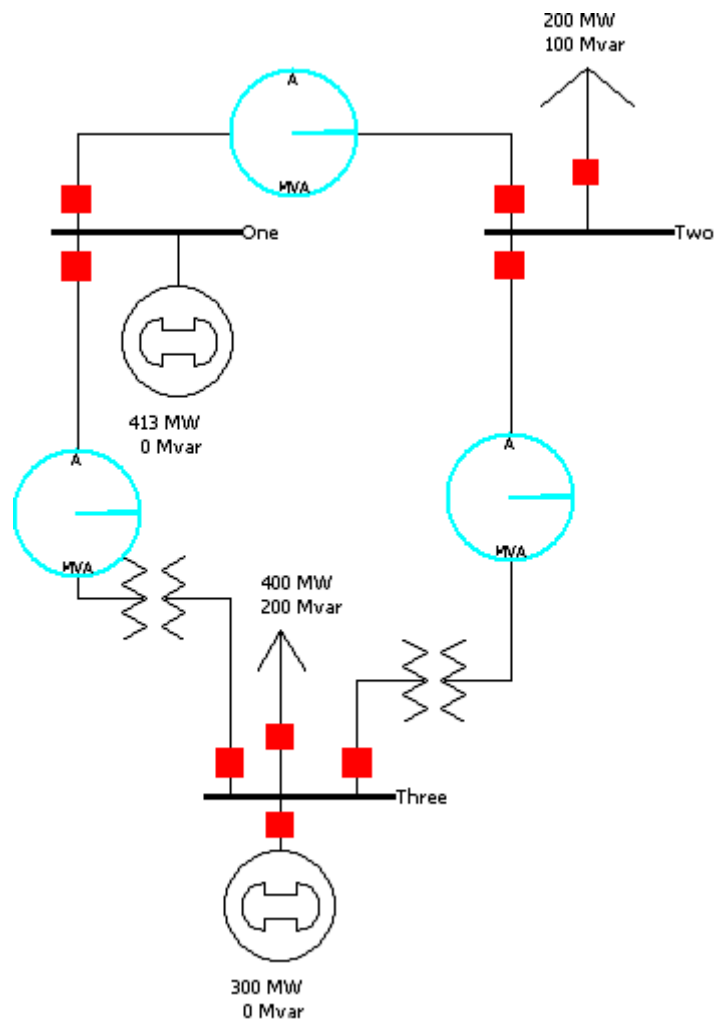
Note that Lines, Loads, Interfaces, Generators and Switched Shunts can all be inserted as objects on a oneline using the **Auto Insert** tool if a record already exists for the device.

- Left-click and drag the load to the desired location on bus 3. Note: you can resize the bus object on the oneline by left-clicking on the bus then dragging either end-point vertex to the desired bus size.
- Right-click on the load and select **Load Information Dialog**. You can change the load orientation and verify all load parameters in this dialog.
- Click **OK**.

Next we are going to attach a 300 MW generator to the bus 3:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click on bus 3. The [Generator Option Dialog](#) opens.
- Select the *Power and Voltage Control* tab. Enter '**300**' in the *MW Output* Field.
- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator appears on the oneline attached to bus 3.

Your oneline should now resemble the image shown below.



Switched shunts usually consist of either capacitors to supply reactive power (in MVAR) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps. If at least one block is in service, the shunt is said to be online. The shunt's corresponding circuit breaker is used to determine and / or toggle the switched shunt's status.

- To insert a switched shunt at bus 3, select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on or near bus 3. The *Switched Shunt Options* dialog box appears (pictured below).
- Verify that the bus number is 3; if it is not, change it.
- Enter **10** for the *Nominal Mvar*.
- Click **OK** to accept the default values of the remaining fields, close the dialog, and insert the switched shunt.

Switched Shunt Options

Bus Number: 3 Find By Number

Bus Name: Three Find By Name

Shunt ID: 1 Find ...

Labels ...

Status: ☐ Open ☒ Closed

	Number	Name
Area	1	1
Zone	1	1
Substation		

Display Parameters Control Parameters Fault Parameters Custom

Nominal Mvar:

Control Mode: ☒ Fixed ☐ Discrete ☐ Continuous ☐ Bus Shunt (Fixed)

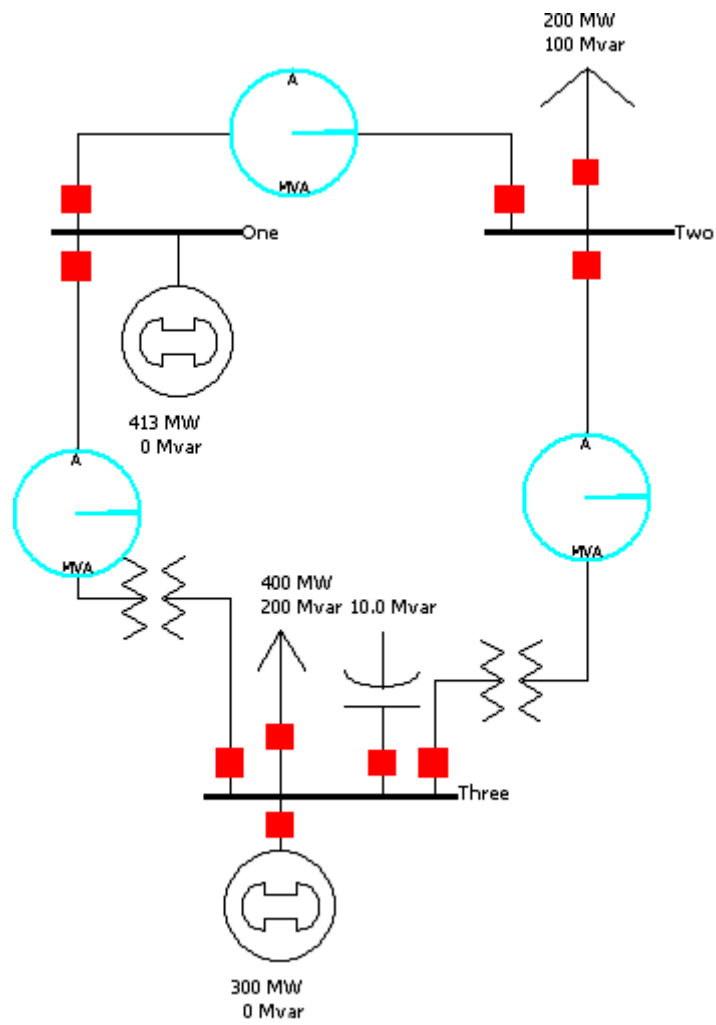
Control Regulation Settings: ☒ Voltage High Value: 1.0000 Low Value: 0.9900 Target Value: 1.0000 Reg. Bus #:

Switched Shunts Blocks

Number of Steps	Mvars per Step

OK Save Cancel Help

Your oneline should now look similar to the image below.



Informational fields can be entered directly on the oneline to allow for ease of monitoring when a case is animating.

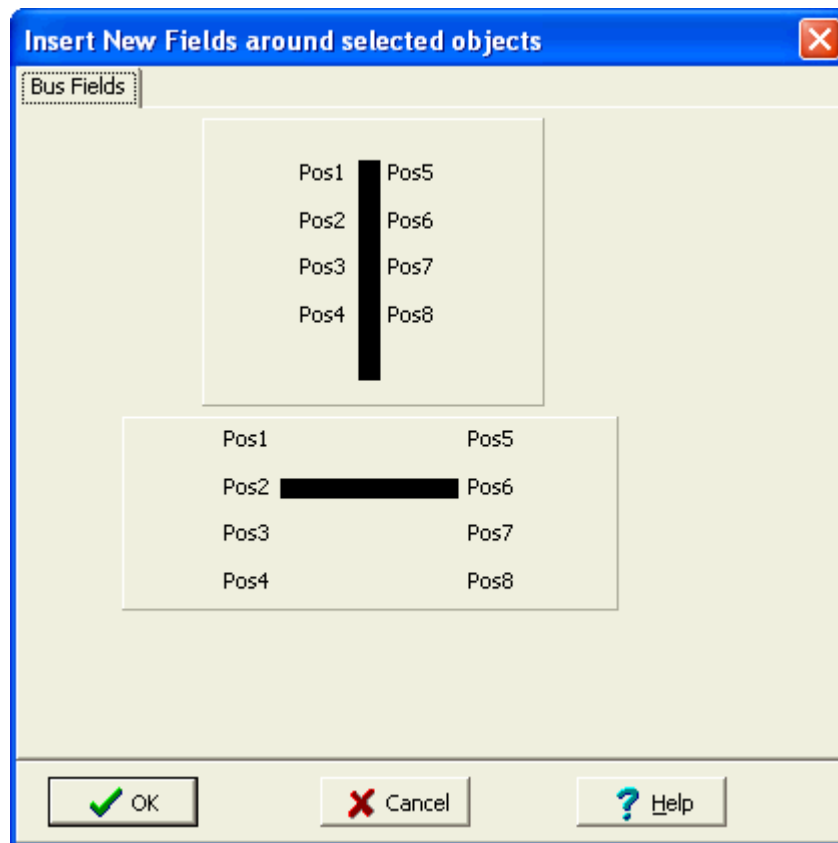
To insert a general text field:

- Select **Background > Text** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Left-click on the oneline in the desired text location to bring up the *Text Object Dialog*. (For this example, left-click in the top center of the oneline.)
- Type the string "First Case" in the *Enter the text* field. Click **OK**.
- To format the text, ensure the text is selected on the one-line then select ¶ from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This displays the [Font Tab](#) of the [Format Selection Dialog](#).
- Set the font size to **26** and the font color to **blue**.
- To change the text background color, select ¶ from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, which summons the Line/Fill Options Tab . If the format menu is already open, click on the [Line/Fill Options Tab](#) of the Format Selection Dialog. Check the *Use Background Fill* box to give the text a white background, and then click **OK**.
- Inserted text can be moved using the same method as any other object on the oneline

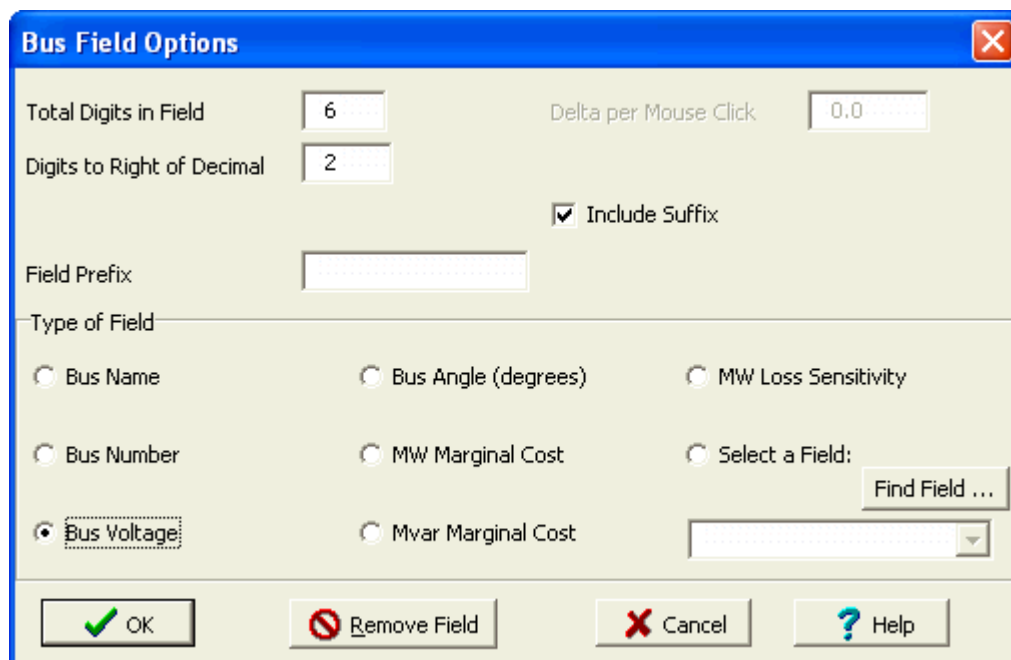
Fields can also display object-related quantities. By default, Simulator has inserted the bus names, generator and load MW and MVAR, and switched shunt MVAR. For this example, we will add a Bus Voltage Magnitude field to each bus and fields showing the power flow on the transmission line and the transformers. Note that object fields can be formatted just like text fields by using the Format menu.

To add additional fields to the display of a particular bus:

- Right-click on the bus to bring up the bus' local menu.
- Select *Add New Fields Around Bus* from the local menu. This opens the [Insert Bus Fields Dialog](#) (pictured below). You may add up to 8 fields per bus. Select the position where you would like to add the new field (position 5 for this example) then click **OK**.



- This opens the Bus Field Options dialog (shown below); select the field **Bus Voltage** to add in the selected position and click **OK**.



- The parameter and position are displayed as highlighted in the Insert New Fields dialog. Click **OK**. Note that the specified bus field has been added to the oneline diagram.
- Repeat this procedure for the other two buses. If necessary, you may move fields manually with the mouse.

Fields can also be inserted using the **Background > Field** option from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Next, we will insert fields showing the power flow at each end of the transmission line.

- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

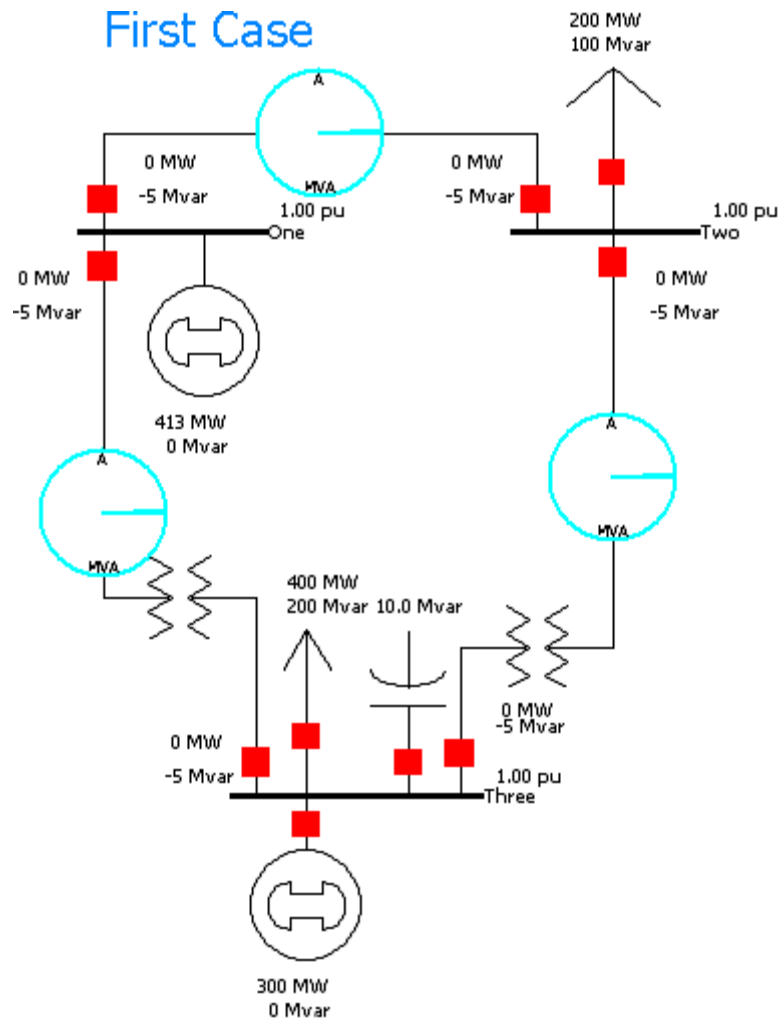
Line fields show information about transmission lines and transformers. For line fields, flow is always specified at an end of the transmission line or transformer. The end is normally determined automatically by the insertion point.

- Left-click near both bus **1** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- The *Near Bus* and *Far Bus* fields should show **1** and **2** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**. The field is displayed on the oneline in the location you specified. Note that the field can be moved and formatted as previously discussed.
- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click near both bus **2** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- Now the *Near Bus* and *Far Bus* fields should show **2** and **1** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**.
- Repeat the procedure to insert **Mvar Flow** fields for the two locations.

We also desire to monitor the MW and Mvar flows on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers. The same commands are used as those used to insert fields for the transmission line.


- Repeat the above steps to insert **MW** (and **Mvar**) **Flow** fields on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers.
- At this point, your first oneline diagram should resemble the one shown below.

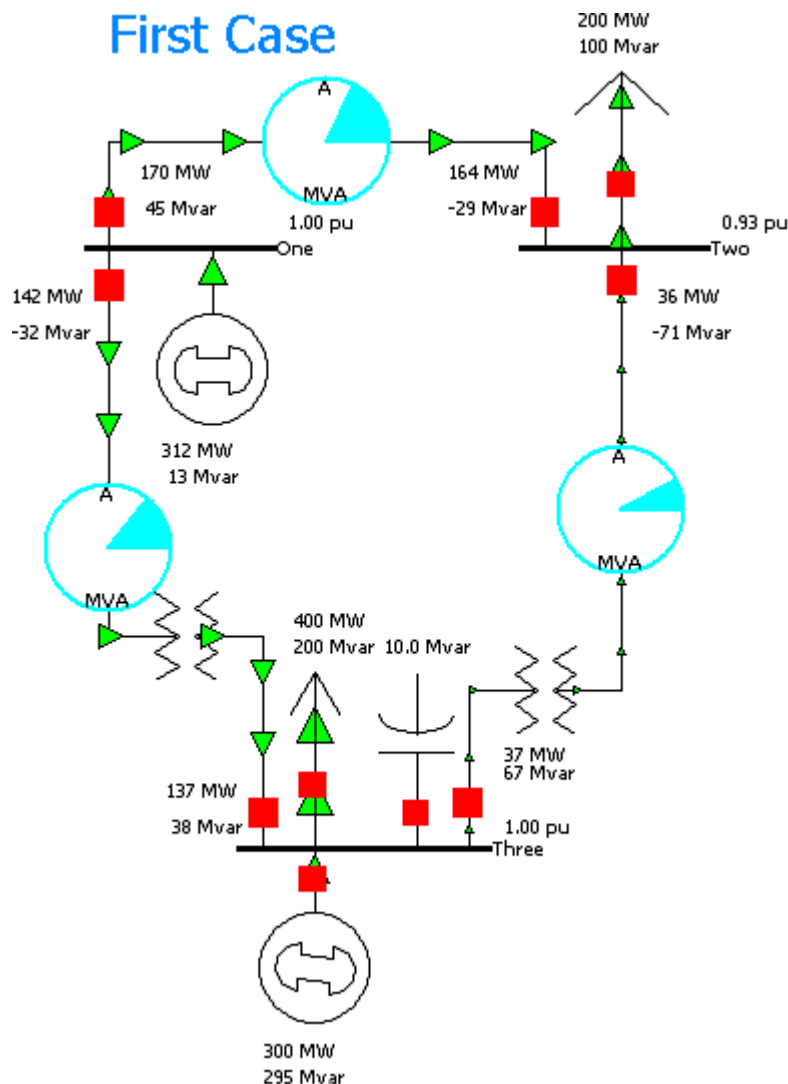
First Case



- Save the case.

To solve a case, you must be in run mode:

- Click on **Run Mode** button in the **Mode** ribbon group. Note that if the case has validation errors a warning will appear. You will need to rectify the problems before you can enter Run Mode. Some validation warning may also appear in the message log.
- Press the **Play**  button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to begin the simulation. Alternatively, to perform a single Power Flow Solution, click the **Single Solution - Full Newton** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Your case should look similar to the case shown below. If it does, congratulations! You have completed building your first case.



Try clicking on the load circuit breaker to toggle the load's status. A solid red circuit breaker indicates that it is closed, a hollow green box indicates it is open. While the simulation is running, click on the circuit breakers and note the nearly instantaneous change in system flows. If the Log window is

visible, you will get a "backstage" view of what Simulator is doing. Feel free to close the log. To re-open the log, click the **Log** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

With the load circuit breaker closed, open the circuit breaker between bus 3 and its connected generator. Now open any of the transmission line or transformer circuit breakers.

Congratulations, you've just blacked-out your case!

Next, we will create a second operating area for the case. Large interconnected systems usually have a number of control areas, with each control area responsible for the operation of a particular part of the system. Often, a single control area corresponds to a single owner (such as an investor-owned utility), but it is not unusual for a single control area to have more than one owner. Control areas are connected to neighboring areas through tie lines. A tie line is a transmission line that has one end in one control area and the other end in another. The total amount of power flowing out of a control area is the algebraic sum of the power flowing out on all the area's tie lines. Each control area is responsible for procuring enough power to meet its own load plus losses. The control area can get this power either by generating it itself, or by buying it from another area. This ability to buy and sell power (i.e., power transactions) is one of the principal advantages of interconnected operation.

To create another control area:

- Change to **Edit Mode**. Right click on bus 3 and select **Bus Information Dialog** from the drop down menu.
- Entering a number for an area that does not already exist automatically creates a new area. Enter '**2**' in the *Area Number* field. Enter '**TWO**' in the *Area Name* field. Click **OK**.
- To verify that the case now has two areas, select **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The [Case Information Displays](#) allow you to view the entire case using non-graphical displays. The [Case Summary](#) dialog (shown below) shows the number of buses, generators, lines/transformers, and control areas in the case. You cannot modify any of these values.

CaseSummary for Current Case

Number of Devices in Case			
Buses	3	Series Capacitors	0
Generators	2	2 Term. DC Lines	0
Loads	2	N-Term. DC Lines	0
Switched Shunts	1	Areas	2
Trans. Lines (AC)	3	Zones	1
LTCs (Control Volt)	0	Islands	1
Phase Shifters	0	Interfaces	0
Mvar Controlling	0	Injection Groups	0

Case pathname: C:\Program Files\PowerWorld\Simulator\Cases\firstcase.PW

Case Totals (for in-service devices only)		
	MW	Mvar
Load	600.0	300.0
Generation	611.7	308.3
Shunts	0.0	-10.0
Losses	11.8	18.3

Generator Spinning Reserves	
Positive [MW]	Negative [MW]
1388.3	611.7

Slack Buses:
One (1); in Area 1 (1)

Buttons: OK, Print, Help

Now we will make sure that both of the control areas are initially set as being on automatic generation control (AGC). AGC insures that the generation in the area is equal to the load plus losses plus and scheduled transactions.

- Select **Case Information > Aggregation > Areas**. The Area display (pictured below) provides a convenient summary of all the control areas in the case. Similar displays exist for buses, generators, lines/transformers, etc.

Area Records								
	Area Num	Area Name	AGC Status	Gen MW	Load MW	Tot Sched MW	Int MW	ACE
1	1	1	Off AGC	311.74	200.00	0.00	99.99	
2	2	Two	Off AGC	300.00	400.00	0.00	-99.99	

- Right-click anywhere on the record for the first area and select the *Show Dialog* option. The [Area Display Dialog](#) is shown (as below).

1 Area Information for Current Case																																																						
Number <input type="text" value="1"/>		<input type="button" value="Find By Number"/>		Area MW Control Options <input type="radio"/> No Area Control <input type="radio"/> Participation Factor Control <input checked="" type="radio"/> Economic Dispatch Control <input type="radio"/> Area Slack Bus <input type="radio"/> Injection Group Area Slack <input type="radio"/> Optimal Power Flow Control																																																		
Name <input type="text" value="One"/>		<input type="button" value="Find By Name"/>																																																				
Super Area <input type="text"/>		<input type="button" value="Find ..."/>																																																				
Labels ... <input type="text" value="no labels"/>																																																						
<div> <div>Info / Interchange</div> <div>Options</div> <div>Area MW Control Options</div> <div>Tie Lines</div> <div>OPF</div> <div>Buses</div> <div>Gens</div> <div>Loads</div> <div>Memo</div> </div>																																																						
Area Summary Totals <table border="1"> <thead> <tr> <th></th> <th>MW</th> <th>Mvar</th> </tr> </thead> <tbody> <tr> <td>Load</td> <td>200.0</td> <td>100.0</td> </tr> <tr> <td>Generation</td> <td>311.7</td> <td>12.9</td> </tr> <tr> <td>Fixed Bus Shunts</td> <td>0.0</td> <td>0.0</td> </tr> <tr> <td>Switched Bus Shunts</td> <td></td> <td></td> </tr> <tr> <td>Line Shunts</td> <td>0.0</td> <td>0.0</td> </tr> <tr> <td>Losses</td> <td>11.75</td> <td>18.25</td> </tr> <tr> <td>Interchange</td> <td>100.0</td> <td>-105.3</td> </tr> <tr> <td>Hourly Cost (\$/hr)</td> <td colspan="2"><input type="text" value="0"/></td> </tr> </tbody> </table>					MW	Mvar	Load	200.0	100.0	Generation	311.7	12.9	Fixed Bus Shunts	0.0	0.0	Switched Bus Shunts			Line Shunts	0.0	0.0	Losses	11.75	18.25	Interchange	100.0	-105.3	Hourly Cost (\$/hr)	<input type="text" value="0"/>		Interchange <table border="1"> <tbody> <tr> <td>Unspecified MW</td> <td><input type="text" value="0.0"/></td> <td>AGC Tolerance</td> <td><input type="text" value="1.00"/></td> </tr> <tr> <td>Transaction MW</td> <td><input type="text" value="0.0"/></td> <td>Area Control Error (ACE)</td> <td><input type="text" value="100.0"/></td> </tr> </tbody> </table> <div> <input type="checkbox"/> Only Show Areas with Nonzero Interchange <input type="checkbox"/> Area Has Multiple Islands </div> <table border="1"> <thead> <tr> <th>Number</th> <th>Name</th> <th>MW Export</th> <th>Enabled</th> <th>ID</th> </tr> </thead> <tbody> <tr> <td>2</td> <td>Two</td> <td>0</td> <td></td> <td></td> </tr> </tbody> </table>						Unspecified MW	<input type="text" value="0.0"/>	AGC Tolerance	<input type="text" value="1.00"/>	Transaction MW	<input type="text" value="0.0"/>	Area Control Error (ACE)	<input type="text" value="100.0"/>	Number	Name	MW Export	Enabled	ID	2	Two	0		
	MW	Mvar																																																				
Load	200.0	100.0																																																				
Generation	311.7	12.9																																																				
Fixed Bus Shunts	0.0	0.0																																																				
Switched Bus Shunts																																																						
Line Shunts	0.0	0.0																																																				
Losses	11.75	18.25																																																				
Interchange	100.0	-105.3																																																				
Hourly Cost (\$/hr)	<input type="text" value="0"/>																																																					
Unspecified MW	<input type="text" value="0.0"/>	AGC Tolerance	<input type="text" value="1.00"/>																																																			
Transaction MW	<input type="text" value="0.0"/>	Area Control Error (ACE)	<input type="text" value="100.0"/>																																																			
Number	Name	MW Export	Enabled	ID																																																		
2	Two	0																																																				
<div> <input checked="" type="button" value="OK"/> <input type="button" value="Save"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/> <input type="button" value="Print"/> </div>																																																						

- Change the Area *Name* to '**One**' and set the AGC Status under *Area MW Control sOptions* to **Economic Dispatch Control**. Select **SAVE** to save this information.
- Next click on the up arrow next to the right of the Area Number field. This displays the Area Record dialog for the next area. Set the AGC Status to **Economic Dispatch Control**, and select **OK** to save your changes and close the dialog.
- Note the *AGC Status* field in the **Area Records Display** now shows **ED** in the *AGC Status* fields. Close the Area Records display.

This procedure describes how to create a power system model from an existing power flow file using PowerWorld Simulator. This tutorial assumes that you have at least some familiarity with PowerWorld Simulator. If you need a more general introduction, please see [Creating a New Case tutorial](#).

PowerWorld cases can be easily created from existing power flow cases stored in PTI RAW versions 23-30, GE PSLF text format (EPC version 11.X and later), and IEEE common format.

Simulator provides a static model of a power system. The power flow data is a subset of the system model. For some studies this model is sufficient. For other studies the model needs to be augmented by adjusting: generator cost information, the reactive capability curve, PowerWorld Simulator case options, interface definitions, injection group definitions, contingency definitions, the time variation of the load, etc....

To begin, double-click on the PowerWorld Simulator icon. This starts Simulator. In this example we will be building a case from an existing power flow file.

Once Simulator has been started, set Simulator to Edit Mode by clicking on the **Edit Mode** button. Select **Open Case** from the [Application Button](#). An **Open** dialog box will appear on the screen. To select a power flow file, click on the **Type of File** field in the lower left hand corner of the dialog box. A list of file formats will appear. Choose the appropriate format, and the available files of the type selected will appear in the box above **Type of File**. Choose the desired file from the list of available files and select *OK*. *In this example, we will be building a oneline for the 3990 bus MAIN 1998 summer case saved in PTI version 23 format as mdb98s.raw. When prompted to create a oneline, select 'yes'.* Your display should immediately turn white. This shows the blank background upon which you will be drawing the oneline diagram for your case. Onelines are used in power system analysis to represent the actual three-phase power system using a single line to represent each three phase device. The information from the power flow file is now accessible to Simulator.

Simulator presents many different [Case Information Displays](#) to provide a text-based view of the case. For example, to view or modify the description of the case, select **Case Description** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The **Case Description Dialog** should appear, showing the text description of the power flow case. In PowerWorld, this text description may be arbitrarily long.

You can also obtain a case summary by selecting **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The summary dialog will appear. It provides a summary of the case and of the total case load and generation. None of the fields in this dialog can be changed, as they are intended for informational purposes only.

The Area/Zone Filters information display is another important article. The [Area/Zone/Owner Filters](#) feature restricts the contents of other case information displays to certain areas or zones. This is particularly useful for large cases. Open the Area/Zone Filters display by selecting **Area/Zone Filters** from the **Case Information** ribbon group on the [Case Information](#) ribbon tab. As with all Case Information Displays, you can click on a column heading to sort the list on a particular field; click on that same column heading again to reverse the sort order. To change a particular area's area/zone filter status, simply left-click on the area's area/zone filter status field. To change the filter status for all areas in the case, right-click on the display to bring up its local menu (all case information displays have a local menu), and select either **Toggle All Yes** or **Toggle All No**. For our example, set all areas to 'no' except WUMS.

Another important text-based display is the Power Flow List. The [power flow list](#) shows the complete power flow information for all areas whose area/zone filter is set to 'yes'. To display the power flow list, select **Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The power flow list has other options that can be accessed by right-clicking anywhere on the list. If you wish to view more details about a particular device, you can do so by holding down the **Ctrl** key and left-clicking on the device. To move through the list of buses, you can use the arrow keys or the scroll bar. If you are viewing a particular bus and you wish to view a bus connected to it, double-click on the line connecting the two buses to view the flows at the other bus. Note that some of the transmission lines have a circuit identifier of '99'. Whenever a '99' is used for an identifier, it usually means that the transmission line is an equivalent line.

A shorter version of the power flow list can often be more useful. Such a display is offered by the Quick Power Flow List. The [Quick Power Flow List](#) is similar to the power flow list except that it shows flows at individual buses or a set of buses, regardless of area/zone filter settings. You can view flows at any bus in the case. To display the quick power flow list, select **Quick Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. Identify the buses of interest in the **Bus Number** field by entering the number of either a single bus, a set of buses separated by commas, or a range of buses specified using a hyphen. Again you can double-click to move to a desired bus, or choose **Show Object Dialog** from the display's local menu to see the information dialog for any object.

To solve the power flow case we have been using in this example, press the **Single Solution - Full Newton** button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab, or press the Single Solution icon on the [Quick Access](#) toolbar. You may wish to show the Message Log before you solve the case so as to monitor the solution process. When you choose to perform a Single Solution, the application automatically switches to Run Mode if it is not already there. The system has initial mismatches because of voltage truncation in the power flow file. The case should converge quickly, perhaps in 2 or 3 iterations. After the single solution has been performed, Simulator now has the solved power flow in memory, and you are ready to build the oneline.

It is not necessary that the power flow case be solved before you create the oneline. However, we recommend that you solve the existing case first to make sure that it is valid.

PowerWorld Simulator makes the power system case easier to analyze by presenting results visually using a oneline diagram. You do not need to represent every bus in the power flow model on the oneline; a oneline diagram need be created only for the desired portion of a system under study. Simulator can automatically link the constructed oneline diagram to the existing power system model. The following sections of this tutorial will discuss placing various visual components onto a oneline diagram. In all of the following sections the actions described will relate to the aforementioned MAIN example.

Tutorial: Entering a Bus Page 6 of 15

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices connect together. To build a oneline diagram, you draw the buses, attach devices such as generators and loads to the buses, and connect the buses together with transmission lines and transformers.


To begin entering devices onto the blank oneline, you must first switch to Edit Mode. If you are not already in Edit Mode, switch back to Edit Mode now. Show the Quick Power Flow List and move it towards the bottom of the screen. Show bus 39820 by typing that number in the **Bus Number** field. Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline towards the top center to define the point at which the new bus will be added. The Bus Information Dialog will appear. In the **Bus Number** field enter 39820. Select **Find by Number** to view the bus information. You should see the information appear in the Bus dialog fields that corresponds with the MAIN power flow case. Select *OK* to place the bus. The bus should now appear on your screen.

If it has not already been done for you automatically, add a bus field identifying the new bus' number immediately to the left of the new bus. To do this, click on or to the left of the display object that represents bus 39820 and select **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. The Bus Field Options Dialog will appear for you to fill out. Designate the type of field as *Bus Number* and close the dialog. Simulator will add a text object showing the bus number at the point where you had clicked.

Repeat this procedure to place buses 39881 and 39821 on the oneline, along with their bus numbers.


Transmission lines between buses can be inserted manually by choosing **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and clicking on the beginning bus. Vertices may be defined along the way by clicking the mouse on the diagram where vertices should appear. However when creating a case from an existing power flow file, you also have the option to insert transmission line display objects automatically. To do this, select **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Accept the default options and click OK. The lines joining the visible buses on the display are automatically added, along with circuit breakers and pie charts, provided those options are set. Simulator will draw only transmission lines that link buses that have already been drawn on the online diagram. If you add another bus to the diagram, you can again auto-insert lines, and Simulator will only insert lines that are not already present on the display.

Two features of Simulator are indispensable when you have a large, detailed oneline: panning and zooming. To pan from side to side or up and down, either use the arrow keys or the scrollbars on the sides of the oneline. To zoom in or out of the oneline, hold the **Ctrl** key down and press the up arrow to zoom in and the down arrow to zoom out. For the example, pan up and insert buses 39819 and 39841, again with bus fields showing their numbers. Again use **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab to add the transmission lines.

Sometimes you may wish to insert background elements on a oneline such as bodies of water or state lines to convey geographic location. To show static background elements on the oneline, select **Background > Background Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the diagram to start the background line, and click at other points on the diagram to add segments. Double click to terminate the background line. If you wish to fill in the area inside the background line or change the background line's color, select the background line and choose **Line/Fill**  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab and make the appropriate choices in the resulting dialog. You may wish to experiment now with drawing background lines or objects and with adding fill color. Additionally, some political and geographic borders can be inserted by selecting **Auto Insert > Borders** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.

Tutorial: Simulating the Case Page 10 of 15

Once you have constructed a oneline diagram showing the area of interest, you can simulate the case. Simulator does not only show the magnitude and direction of flows on transmission lines, but can also *animate* them. To configure the animated flows, switch to Edit Mode and choose **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. This dialog box also allows you to change the size, density, and fill color of the animated flows for easier visualization. At this point, just click *OK*. Next, save your case by selecting **Save Case** from the [Application Button](#) or by pressing the

Save Case  button on the [Quick Access Toolbar](#). If you have not already saved the case, a **Save As** dialog will prompt you to select a name. Both the oneline diagram and the case will use this name.

To begin solving and animating the load flow solution, switch to Run Mode and then press the Play button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. You should now see Simulator modeling the 3990-bus case.

While the animation ensues, you might want to view or change some study parameters. To do this, first pause the animation. (Note that you do not *have* to pause the animation to tweak parameters. You can adjust anything in the case while the animation runs, too.) Then, right-click on any of the objects on the display, and choose Show Dialog from the popup menu. This will bring up the run-time dialog for the object. Many of the parameters on this dialog can be modified, and the new settings will take effect when the animation is restarted.

Several options are available at run time from the oneline diagram's local menu. To call up the menu, right-click on an empty portion of the oneline. The local menu will appear. Use it to print the oneline, save it as a metafile, or copy it to the clipboard. You can also find a particular bus on the oneline, access panning and zooming options, set oneline display options, view information about the power system area in which you clicked, create a contour plot, and use the difference flows activity, all from the oneline local menu.

Tutorial: Area Page 13 of 15

Often, system data is most conveniently displayed by area. To view the Area Records Display, select **Aggregation > Areas** from the [Model Explorer](#). The resulting display summarizes information about all the areas in the case. You can sort the entries by clicking on the column labels.

You can view a report of limit violations by selecting **Limit Monitoring Settings** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. A display showing bus voltage violations, line/transformer violations or interface violations will appear. To show only violations, select **Violating Elements** from the **Elements to Show** options. If the **Use Area / Zone Filters on List Displays** is checked, then the displayed violations will correspond only to areas whose **Shown** field is set to **Yes** on the [Area/Zone/Owner Filters](#) display.

If you wish to see more information on a bus that appears in the violation list, right-click on the bus number and select [Quick Power Flow List](#) from the resulting local menu. If the [Case Information Display](#) is set to refresh automatically, the list of bus voltage violations will update as new violations occur.

In addition to the few displays discussed in this tutorial, Simulator offers many other [Case Information Displays](#). In most cases, you can view information about buses, generators, lines, transformers, loads, and zones simply by right-clicking on the object in question and choosing the appropriate option from the object's local menu. The best way to become more familiar with the displays and the information contained in them is simply to play with a Simulator case and oneline. Simulator's interface has been designed to be simple and intuitive. If you run into problems, consult the On-line Help for assistance.

Note: The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package. Please contact PowerWorld Corporation at info@powerworld.com or visit the website at <http://www.powerworld.com> for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. With the optimal power flow (OPF) enhancement, Simulator OPF can also solve many of the system control equations using an Optimal Power Flow algorithm. Specifically, Simulator OPF uses a linear programming (LP) OPF implementation.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#), taking into account both [equality and inequality constraints](#). These constraints are used to model the power balance constraints and various operating limits.

In Simulator OPF, the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.

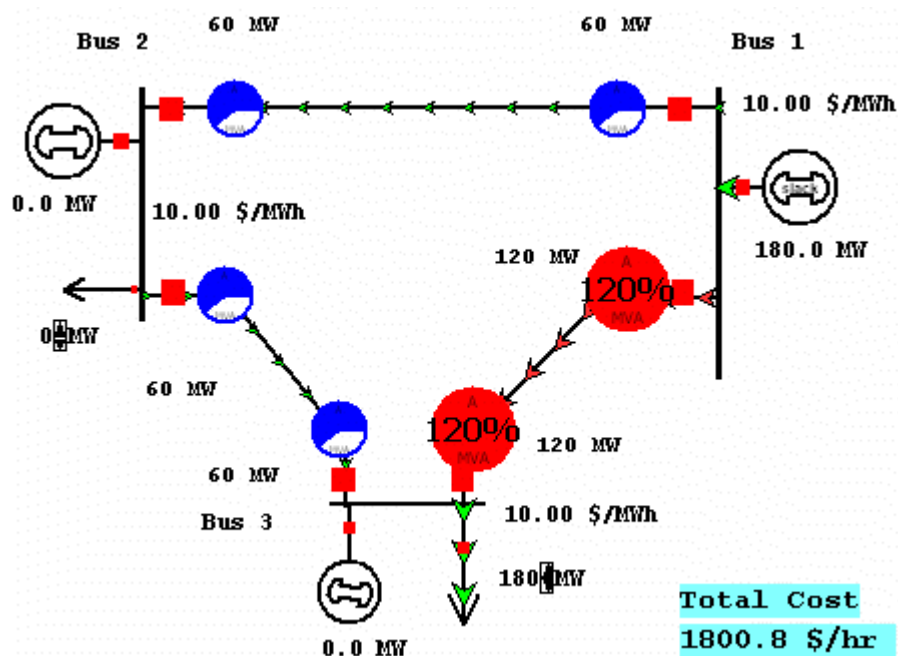
For this tutorial, we will start with a provided three-bus case (B3LP) found in the *PowerWorld\Simulator\Sample Cases* directory. Bus 1 is the system slack bus. All buses are connected via 0.1 pu reactance lines, each with a 100 MVA limit. There is a single 180 MW load at bus 3. The generator marginal costs are:

- Bus 1: 10 \$/MWhr; Range = 0 to 400 MW
- Bus 2: 12 \$/MWhr; Range = 0 to 400 MW
- Bus 3: 20 \$/MWhr; Range = 0 to 400 MW

To begin:

- Load B3LP case.
- Verify the above system specifications by right clicking on each line, bus, and generator and selecting its respective **Line/Bus/Generator Information Dialog** from the drop down menu.
- Select **Run Mode**.
- To be included in the OPF, all required area AGC status fields must be selected to OPF. To set area AGC to OPF: Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Area Records display automatically opens. Verify the AGC Status field is set to **OPF**. If it is not, double click on the field to change its value. Close the OPF Area Records display.
- Select **Add Ons > Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to solve the case. Note: line limits are not initially enforced.

Your display should look similar to the following:



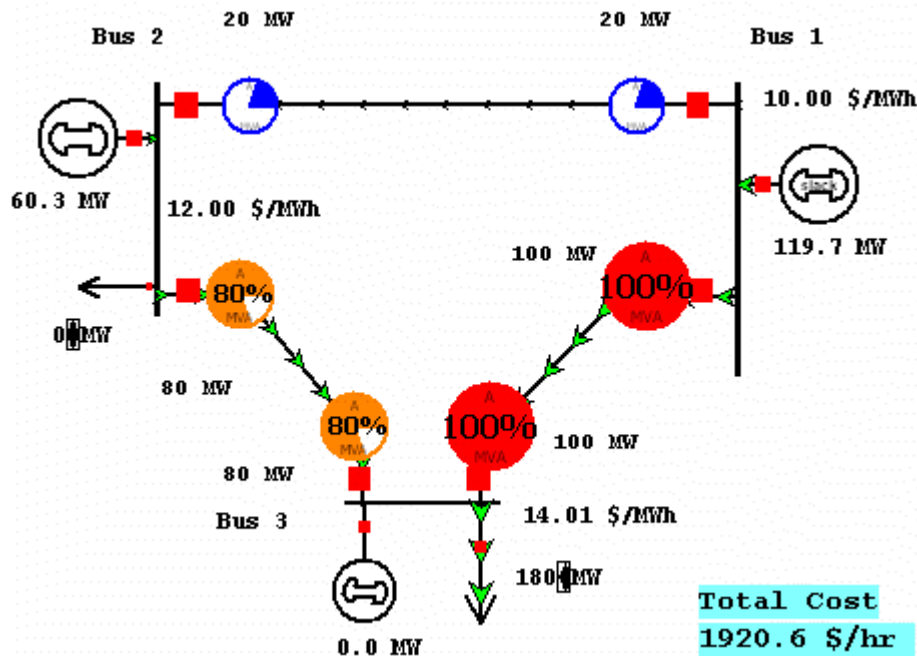
B3LP Solved Using Add Ons > Primal LP

Note the line from Bus 1 to Bus 3 is overloaded and that all buses have the same marginal cost.

- Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

The *Branch MVA* column specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included in the OPF constraints, *Line/Transformer constraints* must not be disabled on the **OPF Options and Results Dialog**, and the individual line/transformer must be enabled for enforcement on the **OPF Line/Transformer MVA Constraints** display.

- Double click on the *Branch MVA* field to change the value to **YES**. Close the OPF Area display
- Select **OPF Options and Results** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.
- The **LP OPF Dialog** opens automatically. Select the **Constraint Options** tab then click *Disable Line/Transformer MVA Limit Enforcement* to remove the checkmark.
- Click **Solve LP OPF** then click **OK**.



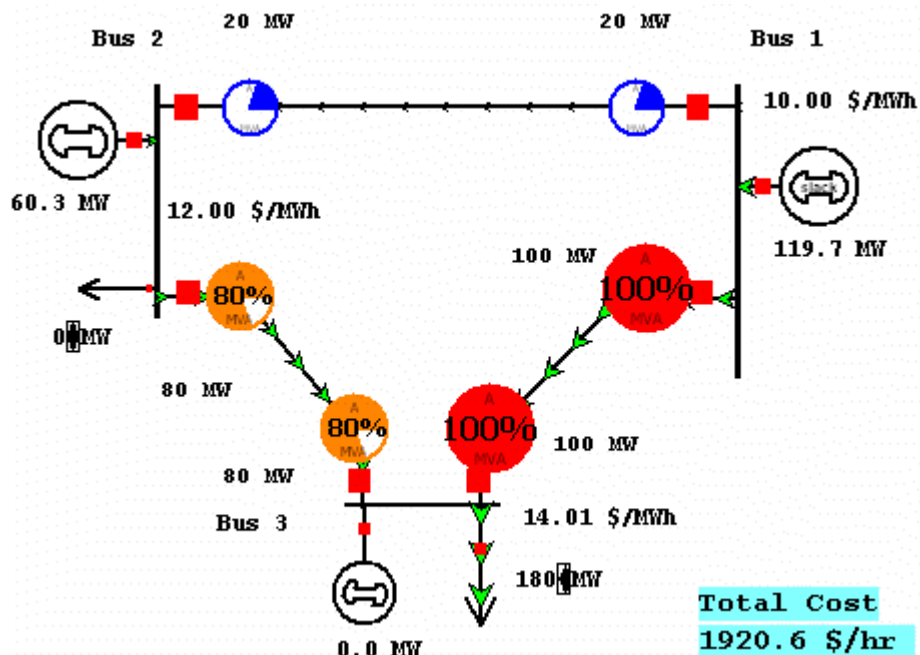
B3LP Solved with Line Limit Enforcement Enabled

The LP OPF re-dispatches to remove the line limit violation. Bus marginal costs have changed from the initial example. The new bus marginal cost at Bus 3 is 14 \$/MWh. To verify this:

- Increase the load at Bus 3 by 1 MW. Right click on the load, enter "181.0" in the *Constant Power / MW Value* field in the **Load Options** dialog. Click **OK**.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

The Total Cost increased to 1935 \$/hr from the previous value of 1921 \$/hr, a difference of 14 \$/hr when Bus 3 load was increased by 1 MW.

Explanation of Bus 3 LMP = 14 \$/MWh (from previous page):



B3LP Solved with Line Limit Enforcement Enabled

All lines have equal impedance. Power flow in a simple network distributes inversely to impedance of the path.

- For Bus 1 to supply 1 MW to Bus 3, 2/3 MW will take the direct path from 1 to 3, while 1/3 MW will take the path from 1 to 2 to 3.
- Likewise, for bus 2 to supply 1 MW to Bus 3, 2/3 will go from 2 to 3, while 1/3 will go from 2 to 1 to 3.
- To supply one additional MW to Bus 3, we need the change in power of generator 1 (Pg1) plus the change in power of generator 2 to equal 1 MW.

$$Pg1 + Pg2 = 1 \text{ MW}$$

- With the line from 1 to 3 limited, no additional power flows are allowed on it.

$$(2/3)Pg1 + (1/3)Pg2 = 0$$

- Solving the above system of equations results in:

$$Pg1 = -1 \text{ MW and } Pg2 = 2 \text{ MW}$$

$$\Delta \text{cost} = \Sigma(\Delta Pg * LMP) = [(-1 \text{ MW}) * (10.00 \text{ $/MWh}) + (2 \text{ MW}) * (12.00 \text{ $/MWh})] = \mathbf{14.00 \text{ $/hr}}$$

Similar to the bus marginal cost, you can also calculate the marginal cost of enforcing a line constraint. For a transmission line, this represents the amount of system savings that could be achieved if the MVA rating was increased by 1.0 MVA.

- Select **OPF Case Info > OPF Lines and Transformers** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Constraints Records dialog opens.

Note the column displaying **MVA Marginal Cost** displays **6.0** for the line from Bus 1 to Bus 3. This is determined based on the following:

- With no change in system load:

$$Pg1 + Pg2 = 0$$

- If we allow one additional MVA to flow on the line from Bus 1 to Bus 3:

$$(2/3)Pg1 + (1/3)Pg2 = 1$$

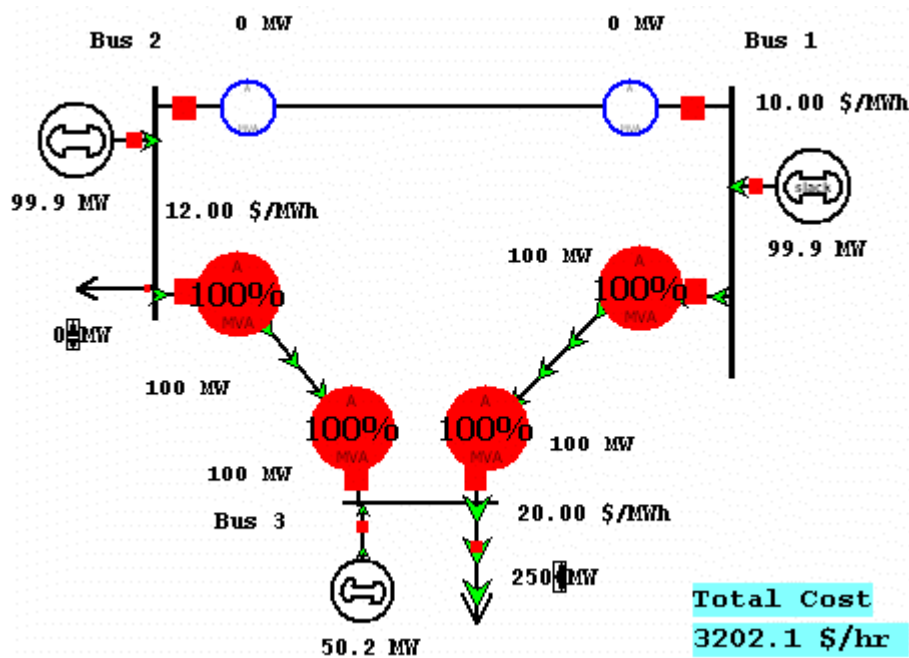
- Solving the above system of equations results in:

$$Pg1 = 3 \text{ MW and } Pg2 = -3 \text{ MW}$$

$\Delta\text{cost} = \Sigma(\Delta Pg * \text{LMP}) = [(3 \text{ MW}) * (10.00 \text{ \$/MWh}) + (-3 \text{ MW}) * (12.00 \text{ \$/MWh})] = \textbf{-6.00 \$/hr}$, a net savings of 6.00 \\$/hr.

Next we will consider a case with unenforceable constraints. To begin, increase the load at Bus 3 to 250 MW then run the LP OPF again:

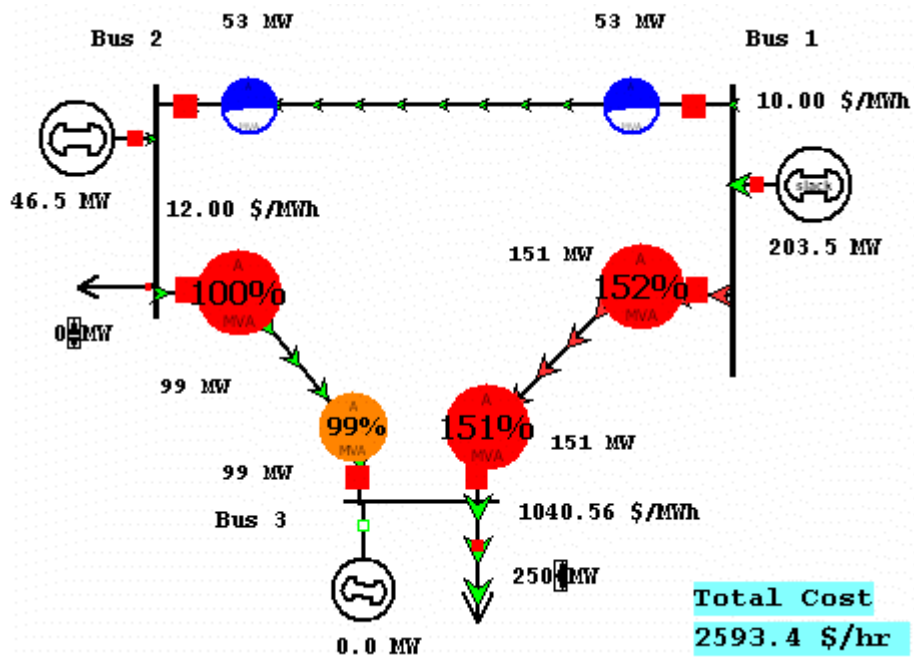
- Right-click on the load at Bus 3. Enter 250 in the Constant Power / MW Value field.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



B3LP with Bus 3 Generator Carrying Load

Note that the transmission lines connecting Bus 3 to the other buses are both at their respective MVA limit and that the generator at Bus 3 is supplying the load in excess of the transmission line limits. Next we will open the generator at Bus 3 thereby inserting an unenforceable constraint:

- Left-click on the red breaker symbol connecting the generator to Bus 3 to open the breaker.
- Select **Primal LP LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



B3LP Solved with Unenforceable Constraints

Both constraints cannot be enforced. If a constraint cannot be enforced due to insufficient controls, the slack variable associated with enforcing that constraint *can not* be removed from the LP basis.

Note the new LMP value for bus 3 exceeds 1000 \$/MWh. The marginal cost depends on the arbitrary cost of the slack variable. This value is specified in the *Marginal Violation Cost* field of the **Add Ons > OPF Options and Results** dialog. See [OPF Options and Results dialog](#) for more details.

Additional Example

To see another OPF example select **PowerWorld Simulator Add-on Tools > Optimal Power Flow (OPF) > Examples** from the table of contents in the on-line help file.

Tutorial: Contingency Analysis

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

1 2 3 4 5 6 7 8 9 10 11

Next

A

Abort, 30, 57	Anchors, 689, 816
Absolute Value, 827	Angle Smoothing, 672
ACE, 220, 222, 676, 679, 940, 942	Animated Flows, 831, 840
Actions, 489, 509, 515, 523, 527, 531, 533, 534, 536, 538, 540, 622	Animation, 820, 831, 840, 975
Add-On Tools, 1180, 1247, 1302, 1384	Appending, 20, 959
Admittance, 97, 285	Case, 20
Advanced, 152, 155, 158, 160	Appending a Case, 959
Advanced Filter, 142, 152, 154, 155, 157, 158, 160, 161, 176	Apply Template, 824
Advanced Filters View Logical Graphical Display, 157	Area, 2, 53, 153, 220, 222, 226, 308, 401, 426, 432, 433, 434, 435, 436, 437, 680, 682, 683, 702, 705, 738, 795, 801
Advanced Sort, 133, 138	Zoom, 53, 801
AGC, 56, 220, 222, 233, 639, 640, 649, 676, 940, 942	Area Average Cost, 683
Aggregation, 42	Area Chart, 680
Aggregations, 111	Area Control, 676
Align Group Objects, 815	Area Control Error, 676, 679
Anchored, 40, 689, 804, 814, 829	Area Display, 220, 222
Anchored Objects, 689, 814, 816	Area Display Objects, 694
	Area Display Options Dialog, 695
	Area Fields, 437, 702, 705, 809
	Area Information, 426, 428, 429, 430, 431, 432, 433, 434, 435, 436

- Area Buses, 428
- Area Gens, 429
- Area Loads, 430
- Area MW Control Options, 431
- Area Loss Sensitivity, 240
- Area Losses, 681
- Area MW Control Options, 431
- Area MW Transactions, 682
- Area Slack, 677
- Area Slack Bus Control, 677
- Area Transactions, 220, 222, 682
- Area/zone, 801
- Area/Zone/Owner, 153
- Area/Zone/Owner Filters, 33, 138, 142, 153
- Areas, 153, 214, 220, 222, 264, 420, 426, 432, 433, 434, 474, 676, 679, 680, 681, 682, 683
- Associate, 105
- ATC, 531, 552, 984, 1341, 1342, 1344, 1351, 1356, 1361, 1363, 1370, 1371
 - Multiple Scenario, 1342, 1361
- ATC Analysis, 1361
- ATC Analysis Methods - Iterated Linear Step IL, 1377
- ATC Analysis Methods - Iterated Linear Step IL then Full CTG Solution, 1380
- ATC Analysis Methods - Single Linear Step SL, 1376
- ATC Analysis Methods - Solution Methods, 1375
- ATC Dialog Analysis Tab, 1359
- ATC Dialog Result Tab, 1358
- ATC Dialog Options Tab, 1343
- ATC Extra Monitors, 1356
- Augmenting Max Flow Min Cut, 938, 939
- Auto Insert, 736, 782, 783, 785, 786, 787, 788, 789, 790, 979, 981, 1020
 - Borders, 790
 - Contingencies, 1020
 - Display Objects, 736, 783, 785, 786, 787, 788, 789
 - PTDF Directions, 979, 981
- Auto Insert Contingencies, 1102

- Auto Insert Injection Groups, 454
- Auto Size, 137
 - Column Widths, 137
- Auto Updating, 801
- Auto Zoom, 801
- Automatic Control, 340, 370, 408
- Automatic Generation Control, 56, 940
- Automatic Line Tap Dialog, 966
- Automatically Insert, 736, 782, 783, 785, 786, 787, 788, 789, 981, 1020
 - Borders, 790
 - Contingencies, 1020
 - Display Objects, 736, 783, 785, 786, 787, 788, 789
 - PTDF Directions, 979, 981
- Autosize, 137
- Auxiliary File
 - Export Format Description, 189
- Auxiliary Files, 75, 88, 89, 91, 92, 93, 484, 1023, 1453
 - Formats, 88, 89, 91, 92, 93, 1023
- Available Contingency Actions, 1009
- Available Transfer Capability, 22, 552, 984, 1341, 1342, 1361, 1371
- Average Cost, 683
- AVR, 230, 233, 340, 370, 408

B

- Back, 41, 805, 810
 - Send, 41, 805, 810
- Background, 41, 42, 746, 748, 749, 750, 800, 801, 805, 810, 811, 827
 - Color, 800, 827
 - Pictures, 750
 - Shapes, 746, 748, 749
- Background Objects, 746, 748, 749, 750, 754, 755, 757, 809
- Background_Lines_Dialog, 747
- Base, 41, 368, 805, 810, 1374
 - Level, 810

Base Case, 478, 479, 480, 482, 903, 904, 905, 907, 918, 919, 920, 922, 1374

Base Case Violations, 1056

Bitmap format, 824

Blink, 828

BMP, 825

Branch, 264, 801, 827, 1371

Object, 801

Branch Limiters, 1371

Branches, 219, 915

Branches that Create Islands, 934

Breakpoint, 450

Bring, 151, 795, 810

Front, 41, 805, 810

Browse, 951

Browse PWB File Header, 950

Browsing, 828

Build, 83, 84

Building New Case, 83

Bus, 2, 38, 40, 228, 229, 230, 286, 289, 291, 387, 470, 473, 689, 707, 708, 709, 793, 801, 804, 817, 821, 827, 839, 929, 957, 960, 961, 962, 969, 970, 972, 1050, 1173

Object, 801

Bus Field Information, 289

Bus Fields, 289, 708, 809

Bus Information, 361, 387

Bus Information Dialog, 137

Bus Load Throw Over Records, 1050

Bus Mismatches, 229

Bus Options, 286

Bus Renumbering, 969, 970, 972

Bus View, 148, 470, 473

Bus Voltage Regulating Devices, 287, 292, 389, 390

Buses, 53, 214, 219, 280, 286, 291, 292, 361, 387, 390, 420, 470, 473, 474, 689, 707, 782, 929, 960, 961, 962, 964, 969, 1173

Buyer, 1342, 1356

C

Calculate Participation Factors, 940, 942

Calculated Fields, 126

Cascade, 68, 877

Case, 20, 22, 71, 76, 78, 79, 82, 85, 137, 212, 214, 821, 959

Appending, 20, 959

Case Description, 212

Case Formats, 71, 76, 78, 98, 99, 212

Case Information, 20, 23, 117, 118, 129, 133, 134, 137, 152, 202, 213, 216, 218, 219, 220, 222, 229, 243, 274, 663

Case Information Customizations, 118, 128

Case Information Displays, 23, 117, 118, 128, 129, 133, 134, 138, 152, 153, 154, 155, 158, 160, 161, 162, 163, 164, 165, 172, 174, 175, 181, 185, 186, 188, 202, 212, 220, 222, 228, 232, 243, 663, 811

Custom Field Descriptions, 122, 124

Case Information Displays Colors and Cell Styles, 131

Case Information Displays Using Cell Handles, 132

Case Information Filterbar, 112, 142, 152, 155, 158, 160, 161

Case Information Toolbar, 112, 118, 137

Case Pathname, 215

Case References, 1013

Case Summary, 214

Case Templates, 824

Case Totals, 215

Center, 53, 801, 871

Display, 53

online, 801

Change, 800

Change button, 809

Change Field Data, 205

Change Fill Color, 809

Change Line Color button, 809

Change Timepoint Time Dialog, 1141

ChangeParametersMultipleElement Function, 1396

ChangeParametersMultipleElement Sample Code, 1397
 ChangeParametersMultipleElementFlatInput Function, 1399
 ChangeParametersMultipleElementFlatInput Sample Code, 1400
 ChangeParametersSingleElement Function, 1393
 Charts, 679, 680, 681, 682, 683
 Check Generator VAR Limits, 56
 Choosing, 810
 Layer, 810
 Circuit, 246
 Circuit Breakers, 719
 Circulating Flows, 930
 Circulating Mvars, 930
 Close Online, 87
 Closest Facilities to Point, 870
 Color, 40, 798, 800, 804, 809, 827, 832, 894
 Color Dialog, 800, 809
 displays, 809
 Color Key, 186
 Color Map, 894
 Colors, 186, 806, 808, 809
 Column Metrics, 151
 Column Widths, 137
 Auto Size, 137
 Comments, 24
 Complete Show, 827
 Conditional Display, 812, 820, 879
 Conditional Display by Zoom Level, 812
 Configuring, 138
 Case Information Displays, 138
 Connect, 1387
 Container Window, 17, 23, 67, 876
 Context-sensitive, 15
 Contingencies, 1019, 1093, 1504
 Contingency, 169, 1011, 1012, 1013, 1020, 1023, 1024, 1026, 1029, 1030, 1031, 1032, 1035, 1038, 1040, 1042, 1045, 1046, 1049, 1051, 1052, 1053, 1054, 1055, 1064, 1065, 1066, 1067, 1068, 1070, 1072, 1074, 1075, 1077, 1080, 1095, 1096, 1097, 1098, 1342, 1669
 Blocks, 1064
 Case References, 1013
 Compare, 1033
 Delete, 1032
 Global Actions, 1028, 1064
 Records, 1013, 1026
 Report, 1033
 Submode, 527
 Contingency Action, 1011
 Contingency Actions, 1032, 1038, 1042, 1070
 Compare Contingencies, 1032
 Contingency Analysis, 2, 168, 266, 984, 994, 1008, 1011, 1012, 1013, 1019, 1020, 1023, 1026, 1030, 1031, 1032, 1035, 1038, 1040, 1042, 1045, 1046, 1056, 1066, 1067, 1070, 1072, 1074, 1075, 1077, 1080, 1095, 1096, 1097, 1098, 1342, 1669
 Advanced Limit Monitoring, 1055
 Advanced Modeling, 1064
 Auto Insert Contingencies, 1020
 Compare Contingencies, 1032
 Contingency Monitoring Exceptions Dialog, 1060
 Custom Monitors, 1062
 Global Actions, 1028
 Load Contingencies, 1023
 Monitoring Exceptions, 1058
 Options, 1045, 1046
 Reference State, 1013
 Saving Records, 1026
 Summary, 1074
 Contingency Analysis Defining Contingencies, 1019
 Contingency Analysis Definitions, 1011, 1064
 Contingency Analysis Results, 1033, 1068, 1069, 1075
 Contingency Analysis Tutorial, 1097, 1098, 1669
 Contingency Blocks, 1029, 1064
 Contingency Case References, 1013, 1017, 1018

- Contingency Case References - State Information, 1014
- Contingency Definition, 1028, 1029, 1035, 1038, 1064, 1068, 1070
- Contingency Element, 1011, 1093, 1504
- Contingency Element Dialog, 1029, 1038, 1070, 1080, 1093, 1095, 1504
- Contingency Records Auto Insert, 1101
- Contingency Specific Options, 1012
- Contingency Violations, 1040, 1077
- Contingency Violations Display, 1036
- Continuously Update, 186
- Contour, 185, 186, 824, 886, 887, 888, 892, 894
- Contour Column, 151, 185, 186
- Contour Type, 186, 888, 892
- Contouring, 129, 185, 186, 886, 887, 888, 892, 894
- Contouring Options, 887, 892, 894
 - Color Key, 894
- Control Areas, 214
- Control Loop, 56
 - Iterations, 56
- Controller Oscillations, 56
 - Prevent, 56
- Controls, 310, 943
- Converge, 23, 640, 672
- Convergence, 23, 56
- Convergence Tolerance, 56, 640
- Converters, 362
- Converting Background Ellipses, 752
- Converting Background Lines, 751
- Converting Background Rectangles, 753
- Coordinates, 871
 - Screen, 871
- Copy, 23, 38, 140, 180, 793, 817
- Copy All, 140, 180
- Copy All menu, 140
- Copy Command, 38, 793, 817
- Copy Online, 841
- Copy Selection, 140, 180

- Copy Selection menu, 140
- Copy Window, 841
- Copying, 841
- Copying Simulator Data, 140, 180
- Cost, 89, 235, 236, 237, 239, 241, 305, 309, 398, 683
- Cost Curve, 89, 235
- Cost Information, 305, 309, 398
- Cost Models, 241
- Counter Flows, 975
- Create Equivalent, 953, 954
- Creating, 19, 107
 - New Case, 19
 - New Project, 107, 108
- Creating Injection Groups, 453
- Criteria Dialog, 795
- Cross Compound Units, 90
- CTG, 1097, 1098, 1669
- Current Zoom Level, 801, 812
- Curves, 235, 312
- Custom, 202, 894
- Custom Case Information Display, 202, 205
- Custom Case Information Displays, 128, 203, 205, 207, 208, 209
- Custom Color Map, 894
- Custom Detail Dialog, 827
- Custom Display, 205
- Custom Expressions, 149, 162, 163, 164, 169
- Custom Expressions Display, 163
- Custom Hint Values, 822
- Customize, 824, 826, 827, 831
- Cut, 38, 793, 817

D

- Data LimitSet, 586
- DC, 214, 260, 261, 357, 360, 361, 362, 364, 639, 651, 653, 724
- DC Transmission Line, 354, 355
 - Actual Flow, 355
 - Inverter Parameters, 354

- DC Transmission Line Options, 350
- DC Transmission Lines, 351, 353
 - Line Parameters, 351
 - Rectifier Parameters, 353
- Decimal, 137, 808
- Decimal places, 137
- Decrease Decimals, 137
- Default, 691, 795, 800, 801, 803, 810, 811, 812
 - Background Color, 800
 - Drawing Values, 803
- Default Drawing Options, 691
- Default Drawing Options Dialog, 38, 39, 793, 794, 803, 809, 817, 818
- Default Drawing Values, 803, 808
- Default Values, 802
- Define Expression, 149, 162
- Define Fields/Strings, 203
- Define Layers, 810
- Define Layers button, 810
- Define String Expression, 164
- Defining, 1019
- Defining Contingencies, 1019
- Definitions, 1011
- Delete, 38, 39, 793, 794, 811, 817, 818
 - Object, 39, 794, 818
- Deleting Injection Groups, 456
- Delphi, 1408, 1416, 1425
- Device Filtering, 152, 155, 158, 160
- Dialog, 795, 803
- Difference, 478, 480, 903, 905, 918, 920
- Difference Case, 479, 904, 919
- Difference Flows, 478, 479, 480, 482, 903, 904, 905, 907, 918, 919, 920, 922
- Digits, 808
 - Right, 808
- Directions, 980, 981
- Disable
 - AGC, 56
 - Generator VAR Limits, 56
 - LTC, 56

- Phase Shifters, 56
- Shunts, 56
- Disable Balancing, 640
- Disable Checking Gen VAR Limits, 641
- Disable Logging, 23
- Disabled
 - Why is this field or line disabled, 129
- Display, 39, 794, 812, 814, 818
 - Conditional, 812
 - Refresh, 138
- Display Detail, 827
- Display Grid Lines on Online, 798
- Display Log, 23
- Display Object, 38, 39, 40, 707, 711, 793, 794, 795, 804, 814, 817, 818, 821, 827
- Display Object Properties, 814
- Display Objects, 713, 715, 719, 724, 726, 731, 733, 735, 742, 755, 963, 964
- Display Options, 688, 800, 838
- Display/Column Options, 137
- Display/Size, 40, 804, 806, 814
- Display/Size Tab, 806, 814
- Displays, 117, 157, 177, 470, 796, 801, 809
 - Color Dialog, 809
- Distance, 983
- Distributed Computing, 1065, 1363, 1616
- Document, 745
- Document Links on Onelines, 745
- Drawing Options, 691

E

- Economic Dispatch Control, 676
- ED, 640, 676
- Edit, 20, 38, 39, 685, 793, 794, 795, 798, 803, 810, 811, 812, 817, 818
- Edit Menu, 38, 39, 793, 794, 817, 818
- Edit Mode, 19, 20, 38, 39, 40, 42, 286, 293, 297, 313, 320, 380, 385, 684, 685, 689, 793, 794, 803, 804, 810, 811, 812, 817, 818, 824
- Edit Mode Introduction, 20
- Edit Mode Overview, 20, 684

Edit Screen Layers, 811
Edit Toolbar, 38, 793, 817
Edit Tools, 38, 793, 817
Enter new keyword here, 261
Environment, 638, 658
Equipment Mover, 963, 964
Equivalencing, 953, 954
Equivalencing Display, 957
Equivalents, 953, 957
Equivalents Display, 954
Excel, 140, 180, 1445
Exciter, 1541
Export, 276, 468
Extra Monitors, 1356
Extra Width, 798

F

F1 key, 15
Facility Analysis, 935, 936, 938, 957
Fault, 307, 1172, 1174, 1176, 1179
 Submode, 533
Fault Analysis, 1171, 1172, 1173, 1174, 1175, 1178, 1179
Field, 42, 182, 289, 795, 808
Field Options, 437
Fields, 182, 289, 345, 347, 348, 376, 383, 689, 690, 703, 705, 708, 714, 716, 718, 728, 732, 734, 737, 754, 757, 758, 945
 Key Fields, 182
File, 70, 103, 215
File Management Options, 666
File Menu, 70
File options, 23
 Save Log, 23
Fill Color, 806, 809
Filter, 33, 138, 142, 152, 153, 154, 155, 158, 160, 161, 176, 647, 945
Find, 134, 137, 172, 801
Find Branches that Create Islands, 934
Find Dialog, 172
Find Records, 134

Finding Records, 134, 172
Fixed Screen Size, 810
Flow Cycles, 930
Flow Sensitivities, 1000
Flow Visualization, 831
Flowgates, 740, 741
Flows, 738, 826, 827, 831, 1000
Font, 23, 40, 804, 806, 808, 810
 choosing, 808
Font Properties, 808
Font Tab, 40, 804, 806, 808
Form Control, 129
Format, 40, 71, 75, 76, 78, 98, 99, 137, 796, 804, 806, 808, 810, 811, 812, 814, 879, 1024
Format Field Properties, 813
Format Menu, 811
Format Multiple Objects, 806, 879
Format Multiple Objects dialog, 810, 812, 814
 Levels/Layers Tab, 812
Format Multiple Options dialog, 806
 open, 806
Format Selection Dialog, 796, 806
 Font Tab, 40, 804
 Levels/Layers Tab, 40, 804
 Line/Fill Tab, 40, 804
Format Toolbar, 40, 804
Free Floating Window, 17, 23, 67, 876
Front, 41, 805, 810
 Bring, 810
Fuel Cost, 239
Fuel Type, 239
Full CTG Solution, 1341
Functional Description of Contour Options, 896
Functions, 165
Functions and Operators, 162, 163, 164, 165

G

Gauges, 42
gcd, 89
gcp, 88

- General, 489, 622, 654
- General Script Actions, 22, 489, 622
- Generation Chart, 680
- Generation Shift Factor, 996
- Generation Shift Factor Sensitivities, 996
- Generator, 90, 233, 235, 236, 237, 239, 241, 297, 299, 301, 302, 305, 307, 308, 309, 310, 312, 393, 395, 398, 400, 401, 827, 943, 964, 1174
- Generator Capability Curves Format, 88
- Generator Cost, 233, 237
- Generator Cost Curve Data Options, 89
- Generator Cost Data Format, 89
- Generator Costs, 241
- Generator Cubic Cost, 239
- Generator Display Objects, 713
- Generator Economic Curves, 235
- Generator Field Information, 299
- Generator Fields, 299, 714, 809
- Generator Information, 301, 305, 308, 309, 393, 395, 398, 400, 401
 - Area, 308, 401
 - Costs, 305, 398
 - Owner, 308, 401
 - Zone, 308, 401
- Generator MW Limits, 56
 - Enforce, 56
- Generator MW Marginal Cost, 239
- Generator Options, 297, 302, 307
 - Fault Parameters, 307
 - MW Control, 302
- Generator Piecewise Linear, 241
- Generator Reactive Power Capability Curve, 312
- Generator Records, 233, 1174
- Generator Spinning Reserves, 215
- Generator VAR Limits, 56
 - Check, 56
 - Display, 56
- Generator/Load Cost Models, 236
- Generators, 214, 219, 230, 233, 235, 236, 237, 239, 241, 267, 269, 271, 282, 299, 301, 309, 310, 312, 393, 396, 400, 452, 453, 456, 457, 459, 462, 463, 466, 689, 713, 714, 785, 838, 924, 943, 963, 964, 1095, 1174
- Generic Model Fields, 757
- Geographic Data View, 193
 - Geographic Data View Styles, 200
 - Fields and Attributes, 195
 - General Display Options, 199
- Geography/Coordinates, 857
- Get Column Metrics, 151
- Get Parameters, 1406, 1408, 1410, 1412, 1414, 1416, 1417, 1419, 1422, 1475
- GetParametersMultipleElementFlatOutput Function, 1421
- Getting, 19
 - Started, 19
- GIS, 854, 855, 856, 858, 865, 867, 868, 870
 - Great Circle Distance Calculation, 855
 - Map Projections, 850
- GIS Shapefile Data, 859, 861, 862, 863
 - Control, 859
 - Format, 862
 - Identify, 860
 - Modify, 861
 - Shape List, 863
- GIS Tools, 854, 855, 856, 865, 867, 868, 870
 - Closest Facilities to Point, 870
 - Delete All Measure Lines, 868
 - Export Oneline as Shapefile, 865
 - Great Circle Distance, 854
 - Insert Measure Line, 867
 - Populate Lon,Lat with Display X,Y, 856
 - Shape File Import, 858
- Governor, 940, 941, 942, 1541
- Graph, 679, 680, 681, 682, 939
- Graphic Formats, 95
- Graphics, 96, 135
- Great Circle Distance Calculation, 855
- Great Circle Distance Dialog, 854
- Grid, 151, 188, 798
 - Snap Objects, 798

Grid Metrics, 188
Grid Metrics Dialog, 151
Grid/Highlight Unlinked Objects, 798, 826
Gridline Color, 798
Gridlines, 798
GSF, 996

H

Help, 15
High Zoom Level, 812
Highlight Unlinked Objects, 798
Highlighting, 798
 Unlinked Objects, 798

I

Identify Cross Compound Units, 90
Images, 96, 135
Immobile, 40, 804, 814
Impedance, 326, 368, 369, 414, 1177
Impedance Correction Tables, 414
Import, 276, 468
Import PTI Subsystems Dialog, 461
Include, 1386
Include Suffix, 808
Increase Decimals, 137
Increase/Decrease Decimals, 137
Indication, 42
inf, 92
Info/Interchange, 432
Initialize, 104
inj, 91
Injection Groups, 91, 215, 267, 269, 272, 452, 453, 456, 457, 459, 461, 462, 464, 466, 581, 827
 Import PTI Subsystems Dialog, 461
Injection Groups Format, 91
Insert, 42, 449, 690, 736, 777, 782, 783, 785, 786, 787, 788, 789, 811
 Area, 42
 Area/Zone, 42

Background, 42
Directions, 981
Display Objects, 42
Fields, 42
Hyperlink, 42
Injection Group, 42
Interface, 42
Link, 42
Memo, 42
Picture, 42
Pie Charts, 42
Screen Layer, 811
Text, 42
Zone, 42
Insert Palettes, 777
Install, 1385
Installing, 1385
InterArea, 738
Interchange, 220, 222
Interchange MW, 220, 222
Interface Data Format, 92
Interface Fields, 447, 737
Interface Information, 444, 446, 447, 448
Interface Limiters, 1371
Interfaces, 92, 215, 264, 266, 420, 444, 446, 447, 448, 449, 450, 474, 583, 735, 736, 737, 739, 740, 788, 801, 828, 976, 1371
Introduction, 3
Inverter Parameters, 354
IOA, 239
Island Based AGC, 649
Island Slack Bus, 274
Island-Based AGC, 639
Islands, 215, 274, 649, 940, 942
Iterated Linear Step, 1341, 1372
Iterations, 23, 56, 640, 672
 Control Loop, 56
 Maximum, 56
 Maximum Number, 56

J

Jacobian, 97, 284

K

Key Field, 664

Key Fields, 182

Keyboard Short Cut Actions Dialog, 880

Keyboard Shortcuts, 880

L

Labels, 420, 423, 474, 477

Layer, 40, 795, 804, 806, 810, 811, 812

 Screen Stack, 41, 805

 Stack, 40, 804

Layers, 810, 812, 874

 choosing, 810

Level, 40, 795, 804, 809, 810, 812, 874

 Background, 41, 805

 Base, 41, 805

 Middle, 41, 805, 810

 Top, 41, 805, 810

Levels/Layers, 40, 804, 806, 810, 811, 812

Levels/Layers Options, 810, 811

Levels/Layers Tab, 40, 804, 806, 810, 812

 Format Multiple Objects dialog, 812

Limit, 1341

Limit Groups, 915

Limit Monitoring, 1045, 1046, 1054, 1055

Limit Monitoring Settings, 909, 910, 915, 1341, 1374

Limit Monitoring Settings and Limit Violations Dialog, 910

Limiters, 1371

Line, 38, 39, 40, 246, 253, 283, 783, 793, 794, 795, 804, 806, 809, 817, 818, 827, 966, 1175

Line Color, 809

Line Field Options, 345

Line Fields, 718, 809

Line Flow Gauge Options Dialog, 722

Line Flow Gauges, 721

Line Information, 343, 364

Line Loading Replicator, 1005

Line Outage Distribution Factors, 984

Line Per Unit Impedance Calculator Dialog, 326

Line Shunt Display, 259

Line Shunts, 325

Line Shunts Information, 325

Line Taps, 965, 966

Line Thickness, 809

Line Voltage Profile, 249

Line/Fill, 40, 804, 806, 809

Line/Fill Properties, 809

Line/Fill Tab, 40, 804, 806, 809

Line/Transformer Information, 404

Linear ATC, 1342, 1372

Linear Calculation Method, 1342

Linearized AC, 1342

Lines/Transformers, 214, 1175

Links, 743, 745

 Other Onelines, 743

List of Devices, 1423, 1425, 1427, 1429

ListOfDevicesAsVariantStrings Function, 1431

ListOfDevicesFlatOutput Function, 1432

Load, 236, 243, 313, 317, 319, 402, 839, 1050, 1095, 1178

Load Benefit Models, 244

Load Characteristic, 1541

Load Chart, 680

Load Display Objects, 715

Load Field Information, 317

Load Fields, 317, 716, 809, 1050

Load Flow, 24

Load ID, 243

Load Information, 402

Load Modeling, 319

Load Multiplier, 220, 222

Load NERC Flowgates, 740

Load Options, 313, 315, 316

 Load Information, 315

 OPF Load Dispatch, 316

- Load Relay, 1541
- Load Schedules, 220, 222
- Load State, 1433, 1434
- Load Status, 243
- Load Throw Over, 1050
- Load Variation, 220, 222
- Loading NERC Flowgates, 740
- Loads, 214, 219, 236, 243, 244, 267, 269, 271, 281, 313, 317, 402, 420, 452, 453, 456, 457, 459, 462, 463, 466, 474, 689, 715, 716, 786, 924, 963, 964, 1095, 1178
- Local Menu, 129, 824
- Local Menus, 209
- LODF, 984, 986, 989, 1341
- Log, 23, 30, 57
 - Windows, 23
- Long Line Voltage Profile, 249
- Loss, 997
 - Sensitivities, 997
- Loss Sensitivities, 997
- Loss Sensitivity, 240
- Losses, 653, 681
- LTC, 56
- LTC Transformer, 340, 370, 408
- LTC Transformers, 372, 410

M

- Machine Model, 1541
- Maintain Fixed Screen Location, 810
- Maintain Fixed Screen Size, 810
- Make Default, 808
- Make Default button, 808
- Make-Up Power Sources, 1095, 1096
- Map Projections, 850
- Matrix Grids, 1126
- Max Flow Min Cut, 938, 939
- Maximum, 56
 - Control Loop Iterations, 56
 - Iterations, 56
- Memo, 24, 436

- Memo Display, 24
- Memo Text, 756
- Menus, 70
- Merging Buses, 960
- Message Log, 23, 30, 57, 672
 - Print, 23
 - Save, 23
 - Show, 23
 - Using, 23
- Metrics, 151, 188
- Microsoft Excel, 140
- Middle, 41, 805, 810
 - Level, 810
- Minimal, 827
- Minimal Show, 827
- Minimum Highlighted Object Pixel Size, 798
- Minimum Sensitivity, 640
- Mismatch, 229
- Model Conditions, 589, 647
- Model Conditions Dialog, 175
- Model Conditions Display, 175
- Model Criteria, 647
- Model Explorer, 111, 210
 - Explore Pane, 111
 - Fields Pane, 112
 - Fields Pane, 114
 - Recent Tabs, 112
- Model Expressions, 168, 179
- Model Fields, 757
- Model Filters, 176, 177, 592, 647
- Model Filters View Logical Graphical Display, 177
- Model Object, 821
- Modeling, 175, 176, 236, 237, 241, 244, 319, 940, 1045, 1064
- Models, 175, 237, 647
- Moderate Show, 827
- Modify, 19, 806, 809, 810, 811, 812, 814
 - Orientation, 814
 - Size, 814
- Mouse, 2, 17, 828

- Mouse Wheel Zooming, 828
- Movie Maker, 849
- Multiple Display Fields, 690
- Multiple Scenario, 1371
- Multiple Scenario Analysis, 1342, 1361
- Multiple Scenario ATC Analysis - Results Tab
Local Menu Options, 1369
- Multiple Scenario ATC Dialog Results Tab,
1367
- Multiple Scenario ATC Dialog Scenarios Tab,
1364
- Multi-Section Lines, 219, 253, 343, 597, 725
- Multi-Terminal, 261, 360, 361, 362, 364
- Mutual, 1176, 1177
- Mutual Impedance, 1176, 1177
- MVA flow, 829
- Mvar, 372, 410, 827
- Mvar Control, 372, 410
- MVR flow, 829
- MW, 682, 827, 829, 982
- MW Control, 302
- MW Distance, 982
- MW Marginal Cost, 239
- MW Transactions, 220, 222, 275, 682
- MW/MVR, 827

N

- NERC, 740, 741
- Network, 42, 111
- Network Cut, 945, 946
- New, 83, 84
- New Case, 19, 20, 83
 - Creating, 19
- New Online, 20, 84
- Newton, 64, 65, 669, 670
- Newton-Raphson, 642, 672
- Nomogram Display, 266
- Nomogram Information, 450
- Nomogram Interface Limiters, 1371
- Nomograms, 266, 450

- Number, 56
 - Iterations, 56

O

- O&M, 239
- Objects, 24, 38, 39, 42, 137, 793, 794, 795,
801, 803, 806, 808, 809, 810, 811, 812, 814,
817, 818, 821, 826, 827, 844, 963, 1464
 - Area, 801
 - Area/Zone, 801
 - Background Objects, 746, 748, 749, 750,
754, 755, 757, 809
 - Bus, 801
 - Display, 38, 39, 793, 794, 817, 818
 - Find, 801
 - Zone, 801
- Observing, 810
 - online, 810
- Off-Nominal, 369
- Online, 38, 40, 42, 80, 86, 87, 95, 470, 473,
660, 688, 702, 703, 743, 793, 795, 798, 800,
801, 804, 808, 810, 811, 812, 814, 817, 820,
821, 826, 827, 836, 838, 840, 871, 872, 879,
882, 951
 - Browsing Path, 662
 - Conditional Display, 872, 879
 - Coordinates, 660
 - Diagrams, 80, 688, 692, 702, 703, 705, 707,
708, 714, 716, 719, 720, 800, 820, 827, 882
 - Display, 688, 798, 800, 826, 827, 838
 - Diagrams, 723
 - Display Options, 660
 - Displays, 470, 473
 - Exporting, 95
 - File, 658, 661
 - Font, 661
 - Links, 743, 828
 - Options, 660, 688, 800, 810, 812, 827, 838,
879
 - Save, 660
 - Screen Coordinates, 871
 - Size, 661

- Text Fields, 754
- Visualization, 660
- Online Display Options, 826, 857
 - Geography/Coordinates, 857
- Open, 23, 79, 80, 82, 110, 795, 806, 810
 - Project, 110
 - Screen Layers list, 810
- Open Case, 1435, 1436
- Operators, 165
- OPF, 266, 316, 400, 435, 677, 1008, 1247
 - Constraints, 1249, 1250, 1251, 1253, 1254, 1261
 - Controls, 1248, 1259, 1263, 1275, 1292, 1293
 - Data, 1255, 1258, 1267
 - Example, 1257, 1294, 1297, 1298, 1300
 - Load Dispatch Options, 316
 - LP OPF, 1256, 1270, 1271, 1272
 - Options, 1258, 1259, 1261, 1263, 1267, 1270, 1271, 1272, 1273, 1274, 1275, 1276, 1277
 - Records, 1255, 1278, 1280, 1281, 1284, 1285, 1286, 1288, 1290
 - Trace Solution, 1277
- OPF Control, 677
- OPF DC Lines Records, 1287
- OPF Options Advanced Options, 1266
- OPF Options and Results, 1258
- OPF Reserves
 - Case Information Display, 1336
 - Constraints, 1332
 - Controls, 1330
 - Example, 1337
 - Objective Function, 1334
 - Overview, 1328
 - Reserve Requirement Curves, 1333
 - Reserves Bids, 1331
 - Results and Pricing, 1335
 - Topics, 1329
- Optimal Power Flow, 266, 400, 435, 677, 1008, 1247
 - Constraints, 1249, 1250, 1251, 1253, 1254, 1261
 - Controls, 1248, 1259, 1263, 1275, 1292, 1293
 - Data, 1255, 1258, 1267
 - Example, 1257, 1294, 1297, 1298, 1300
 - LP OPF, 1256, 1270, 1271, 1272
 - Options, 1258, 1259, 1261, 1263, 1267, 1270, 1271, 1272, 1273, 1274, 1275, 1276, 1277
 - Records, 1255, 1278, 1280, 1281, 1284, 1285, 1286, 1288, 1290
 - Trace Solution, 1277
- Options, 23, 56, 137, 638, 639, 640, 649, 651, 654, 655, 658, 663, 673, 688, 691, 795, 798, 800, 801, 803, 806, 811, 812, 827, 983
- Options Toolbar, 56
- Options/Tools, 482, 679, 800, 803, 826, 840, 907, 922, 1000
- Orientation, 806, 814
- Orientation Setting, 814
- Oscillations, 56
- OTDF, 1356
- Other Applications, 180
- Other Display Object Properties, 814
- Out of Service Elements, 660
- Outages, 219
- Out-of-Service Elements, 828
 - Visualizing, 828
- Overview, 103, 452, 462, 684, 776, 820, 935, 1180, 1247, 1302
- Owned, 281, 282, 283
- Owned Bus Records, 280
- Owned Generator Records, 282
- Owned Line Records, 283
- Owned Load Records, 281
- Owner, 153, 278, 279, 280, 281, 282, 283, 308, 401, 600
- Owner Display Objects, 700
- Owner Display Options Dialog, 701
- Owner Fields on Onelines, 706
- Owners, 153

P

- Palettes, 776, 777
- Pan, 2, 53, 801, 810, 872
 - Zoom, 872
- Pan/Zoom, 801, 873
 - New Location, 802
- Pan/Zoom Control, 801, 873
- Panning, 801
- Participation Factor Control, 676
- Participation Factors, 234, 310, 676, 940, 942, 943
- Participation Points, 271, 462, 463, 466
- Paste, 38, 39, 140, 141, 180, 793, 794, 817, 818
- Paste Buffer, 38, 39, 793, 794, 817, 818
- Paste Format Dialog, 807
- Path, 291, 828, 929
 - Browsing, 828
- Pathname, 215
- Pause, 30, 57, 849
- Per Unit, 326, 368
- Phase, 255, 369, 374, 412
- Phase Shift, 369, 374, 412
- Phase Shifters, 56
- Pie Chart / Gauge Dialogs, 764
- Pie Chart / Gauge Example, 775
- Pie Chart / Gauge Style Dialog, 767
- Pie Chart / Gauge Style Dialog - Gauge Parameters Tab, 773
- Pie Chart / Gauge Style Dialog - Open Parameters Tab, 771
- Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab, 772
- Pie Chart / Gauge Style Dialog - Standard Parameters Tab, 769
- Pie Chart Relative, 829
- Pie Chart Style, 829
- Pie Charts, 448, 720, 739, 826, 829
- Pie Charts/Gauges General Options, 763
- Pie Charts/Gauges Interfaces, 761
- Pie Charts/Gauges Lines, 759
- Pies, 42
- Populate Lon/Lat with Display X,Y, 856
- Popup menu, 811
- Post Power Flow, 640, 647
- Post-contingency, 1374
- Potential Misplacements Dialog, 964
- Power Control, 395
- Power Flow, 23, 56, 216, 218, 229, 639, 640, 653, 672, 940, 942, 974, 1050
 - Submode, 523
 - Visualize, 974
- Power Flow List, 216, 218
- Power Flow Solution, 2, 23, 56, 276, 468, 470, 473, 638, 639, 640, 647, 649, 651, 653, 654, 655, 658, 665, 672, 676, 679, 680, 681, 682, 683, 940, 942, 982, 983, 984, 1050
- Power Flow Solution Advanced Options, 643
- Power Flow Solution Message Log, 656
- Power Flow Solution Actions, 640
- Power System Model, 38, 39, 175, 176, 236, 237, 241, 244, 319, 420, 474, 793, 794, 817, 818, 821
- Power System Scaling, 924
- Power Technologies, 1023
- Power Technology, 1024
- Power Transfer Distribution Factors, 974, 975, 1341
- PowerFlow
 - Submode, 523
- PowerPoint, 841
- PowerWorld, 15, 19, 1464
- PowerWorld Corporation, 15
- PowerWorld Project, 103, 104, 105, 107, 108, 110
- PowerWorld Simulator, 15, 19
- Present Case, 478, 479, 480, 482, 903, 904, 905, 907, 918, 919, 920, 922
- Present Topological Differences, 482, 907, 922
- Presentations, 841
- Pre-Transfer Value Estimate, 1374
- Prevent Controller Oscillations, 56, 642
- Print, 882, 883, 885

- Print Message Log, 23
- Print Options, 883
- Print Title Bar, 883
- Print Window, 883
- Printer Setup, 885
- Printing, 882, 883, 885
- ProcessAuxFile, 1437, 1438
- Project, 103, 104, 105, 107, 108, 110
- Properties, 798, 808, 809, 814
 - Display Object, 814
- PSS/E, 1023, 1024, 1026
- PSS/E Load Throw Over Files, 1025
- PTDF, 2, 452, 974, 975, 979, 980, 981, 982, 984, 990, 1341, 1356, 1374
 - Directions, 980
- Publish, 96, 135
- PV, 1180, 1181, 1182, 1203, 1208, 1211, 1215, 1217, 1242, 1244
 - New Plots, 1218
 - Options, 1184, 1185, 1190, 1198, 1200, 1206, 1237, 1245
 - Submode, 534

Q

- Quick Access Toolbar, 29
- Quick Filter, 152
- Quick Power Flow List, 148, 178, 216, 218
- QV, 1215, 1217, 1219, 1221, 1223, 1226, 1228, 1230, 1239, 1240, 1242, 1244
 - Options, 1198, 1206, 1225, 1237, 1245
 - Submode, 536

R

- Range, 178
- Range of Numbers, 178
- Reactive Power Capability, 312
- Recently Opened Cases, 82
- Records, 133, 134, 140, 821
 - Sorting, 129, 133, 138
- Rectangle Selector, 806

- Rectangular Selector, 806
- Rectifier Parameters, 353
- Reference State, 1013
- Refresh, 67, 138, 816, 827, 876
- Refresh Anchors, 816
- Refresh Display, 138
- Regulation, 233
- Relationship Between Display Objects, 38, 39, 793, 794, 817, 818
- Relinking, 798
- Remote Regulation, 230, 233
- Remote Regulation Viability, 672
- Remotely Regulated, 230, 233
- Renumber Areas/Zones/Substations Dialog, 973
- Renumbering, 969, 970, 972
- Report Violations, 1055
- Report Writing, 1067, 1072
- RequestBuildDate Property, 1463
- Required Field, 138, 664
- Required Fields, 184
- Resize, 40, 804, 806, 810
- Resizing, 810
- Restore, 655
- Restore Default Values, 802
- Restore Last Solution, 655
- Restore State, 655
- Ribbon, 26
 - Case Information, 33
 - Draw, 37, 687
 - Clipboard, 38, 793, 817
 - Formatting, 807
 - Individual Insert, 42
 - Quick Insert, 44, 780
 - Select, 46, 797
 - Onelines
 - Active, 49
 - Onelines, 47
 - Onelines
 - Zoom, 53
 - Options, 54

- Quick Access Toolbar, 29
- Tools, 57
- Tools
 - Edit Mode, 59
- Tools
 - Other Tools, 61
- Tools
 - Run Mode, 63
- Window, 67, 876
- Right-clicking, 811
 - online, 811
- Run, 21, 23
- Run Mode, 21, 22, 387, 391, 393, 402, 404, 415, 418, 515, 828, 974
 - Difference Flows, 479, 480, 482, 904, 905, 907, 919, 920, 922
 - Fault Analysis, 1171, 1172, 1173, 1174, 1175, 1178, 1179
 - Governor Power Flow, 940, 941, 942
 - Introduction, 21
 - LODF, 984, 986
 - Options, 941
 - Script, 515
 - Sensitivities, 996, 997, 1000
 - Tools, 168, 235, 940, 950, 974, 984, 990, 996, 1008, 1095
 - Tools & Options, 168, 235, 479, 480, 482, 849, 904, 905, 907, 919, 920, 922, 940, 942, 974, 975, 979, 980, 981, 984, 986, 989, 990, 991, 994, 996, 997, 1000, 1008, 1012, 1013, 1020, 1023, 1024, 1026, 1029, 1030, 1031, 1032, 1035, 1038, 1040, 1042, 1045, 1046, 1050, 1054, 1055, 1065, 1066, 1067, 1070, 1072, 1074, 1075, 1077, 1080, 1095, 1096, 1171, 1172, 1173, 1174, 1175, 1176, 1177, 1178, 1179
- RunScriptCommand, 1439, 1440

S

- Save, 23, 53, 85, 86, 97, 181, 874, 1441, 1443
- Save Case, 181, 1441
- Save Case Information Data, 181
- Save Hidden Layers, 875

- Save Log, 23
- Save View, 53, 874
- Save/Edit/Delete, 874
- Saving NERC Flowgates, 741
- Scaling, 178, 924
- Schedules, 220, 222
- SCOPF, 266, 400, 435, 676, 1008, 1302, 1303, 1304, 1305, 1306, 1311, 1314, 1316, 1317, 1321
 - Constraints, 1307, 1308, 1309
 - Controls, 1311, 1322
 - Example, 1323, 1326, 1327
 - LP, 1316, 1319, 1320
 - LP OPF, 1256, 1270, 1271, 1272
 - OPF Constraints, 1249, 1250, 1251, 1253, 1254, 1261
 - OPF Controls, 1248, 1259, 1263, 1275, 1292, 1293
 - OPF Data, 1255, 1258, 1267
 - OPF Example, 1257, 1294, 1297, 1298, 1300
 - OPF Options, 1258, 1259, 1261, 1263, 1267, 1270, 1271, 1272, 1273, 1274, 1275, 1276
 - OPF Records, 1255, 1278, 1280, 1281, 1284, 1285, 1286, 1288, 1290
- Screen Coordinates, 871
- Screen Layer Options, 812
- Screen Layer Options dialog, 811
- Screen Layers, 811, 812
- Screen Layers list, 810
 - open, 810
- Screen Location, 801, 810
- Screen Size, 810
 - Fixed, 810
- Script Command Execution Window, 22
- Script Commands, 22
- Script General Actions, 22, 489, 622
- Script Mode, 22, 30, 57, 484, 486, 488, 507, 543, 547, 621
 - Actions, 489, 507, 509, 515, 523, 527, 531, 533, 534, 536, 538, 540, 622
 - Data, 554, 555
 - Data, 543, 544, 547, 548, 549, 550, 551, 552

Data, 556
Data, 557
Data, 558
Data, 559
Data, 560
Data, 567
Data, 568
Data, 569
Data, 570
Data, 571
Data, 572
Data, 574
Data, 576
Data, 577
Data, 579
Data, 580
Data, 581
Data, 583
Data, 585
Data, 586
Data, 587
Data, 588
Data, 589
Data, 590
Data, 592
Data, 593
Data, 597
Data, 598
Data, 599
Data, 600
Data, 603
Data, 604
Data, 605
Data, 606
Data, 607
Data, 608
Data, 610
Data, 611
Data, 612
Data, 613

Data, 615
Data, 616
Data, 617
Data, 618
Data, 619
Data for Display Auxiliary Files, 602, 620,
627, 628, 630, 631, 632, 633, 634, 635, 636,
637
Files, 484, 487, 488, 543, 621
Script, 486, 488, 489, 507, 509, 515, 523,
527, 531, 533, 534, 536, 538, 540, 621, 622
Script Mode Introduction, 22
Search, 112, 137, 174
Security Constrained OPF, 266, 400, 435, 676,
1008, 1302, 1303, 1304, 1305, 1306, 1311,
1314, 1316, 1317, 1321
Constraints, 1307, 1308, 1309
Controls, 1311, 1322
Example, 1323, 1326, 1327
LP, 1316, 1319, 1320
LP OPF, 1256, 1270, 1271, 1272
OPF Constraints, 1249, 1250, 1251, 1253,
1254, 1261
OPF Controls, 1248, 1259, 1263, 1275, 1292,
1293
OPF Data, 1255, 1258, 1267
OPF Example, 1257, 1294, 1297, 1298, 1300
OPF Options, 1258, 1259, 1261, 1263, 1267,
1270, 1271, 1272, 1273, 1274, 1275, 1276
OPF Records, 1255, 1278, 1280, 1281, 1284,
1285, 1286, 1288, 1290
Select, 39, 794, 795, 809, 810, 814, 818
Select By Criteria, 795, 806
Select OK, 809
Selected group, 796
Selected objects, 795, 945
Selecting Format, 809
Seller, 1342, 1356
Send, 41, 805, 810
Back, 41, 805, 810
Send All to Excel, 141
Send to Excel, 141

- Sensitivities, 986, 991, 996, 997, 1000, 1032
- Sensitivity Calculations, 1033
- Sequence Data Format, 93
- Series Capacitor, 342, 347, 731
 - Fields, 347
- Series Capacitor Fields, 732
- Set Custom Detail, 827
- Set Generator Participation Factors, 310, 943
- Setting, 800
 - Background Color, 800
- Setting Default Drawing Options, 691
- Shape File Import, 858
- Shapefile, 859, 860, 861, 862, 863
 - Database Record Dialog, 864
 - Export Oneline as Shapefile, 865
 - GIS Shapefile Data Control, 859
 - GIS Shapefile Data Format, 862
 - GIS Shapefile Data Identify, 860
 - GIS Shapefile Data Modify, 861
 - GIS Shapefile Data Shape List, 863
- Shortcuts, 880
- Shortest Path Between Elements, 291, 929
- Show Animated Flows, 831
- Show Fields Primary, 207
- Show Fields Secondary, 208
- Show ThumbNail View, 834
- Shunts, 56, 262, 325, 380, 415, 689, 719, 733, 787, 924, 963, 964, 1179
- SimAuto, 1384, 1385, 1386, 1387, 1389, 1390, 1391, 1393, 1396, 1397, 1399, 1400, 1431, 1432, 1456, 1464
 - ChangeParametersMultipleElement, 1396
 - ChangeParametersMultipleElementFlatInput, 1399
 - ChangeParametersSingleElement, 1393
 - Delphi, 1408, 1416, 1425
 - Functions, 1391, 1392, 1402, 1404, 1406, 1414, 1422, 1423, 1433, 1435, 1437, 1439, 1441, 1443, 1445, 1453
 - Get Parameters, 1406, 1408, 1410, 1412, 1414, 1416, 1417, 1419, 1422, 1475
 - List of Devices, 1423, 1425, 1427, 1429
 - ListOfDevicesAsVariantStrings, 1431
 - ListOfDevicesFlatOutput, 1432
 - Load State, 1433, 1434
 - MATLAB, 1410, 1417, 1427
 - Open Case, 1435, 1436
 - ProcessAuxFile, 1437, 1438
 - Properties, 1456, 1457, 1459, 1461
 - RunScriptCommand, 1439, 1440
 - Sample Code, 1394, 1397, 1400, 1403, 1405, 1408, 1410, 1412, 1416, 1417, 1419, 1425, 1427, 1429, 1434, 1436, 1438, 1440, 1442, 1444, 1446, 1454, 1458, 1460, 1462
 - VB, 1412, 1419, 1429
 - Version 9.0, 1465, 1466, 1467, 1468, 1470, 1471, 1472, 1473, 1474, 1475, 1476, 1477, 1478, 1479, 1480, 1481, 1482, 1483, 1484
- Simulation, 23, 79
- Simulation Control, 30, 57, 64, 669
- Simulator Options, 638
- Single Linear Step, 1341
- Single Solution, 64, 65, 640, 669, 670
- Size, 137, 806, 808, 810, 814, 827
 - modify, 814
- Size/color, 809
- Slack, 215
- Slack Bus, 215, 287, 643
- Slide Presentations, 841
- Snap Objects, 798
 - Grid, 798
- Solution, 23, 56, 229, 276, 468, 472, 473, 638, 639, 640, 647, 649, 651, 653, 654, 655, 658, 665, 672, 677, 679, 680, 681, 682, 683, 982, 983, 1012
- Solution Options, 56, 638, 639, 640, 649, 651, 654, 655, 658, 663, 672, 940, 942, 983, 1012
- Solution Options Toolbar, 56
- Solving, 672
- Sort, 133, 138
- Sorting, 129, 133, 138
 - Records, 140
- Sorting Records, 133
- Special Data Sections, 632

- Spinning Reserves, 215
- Splitting Buses, 961, 962
- Stability, 1180
- Stabilizer, 1541
- Stack, 40, 804, 806, 810
- Stack Level, 40, 804, 806, 810
- Status Bar, 25
- Storage, 639, 655
- Submenu, 825
- Substation, 232, 295, 473, 712, 801, 826, 827, 838
 - Field Options, 295
 - Fields, 712
 - Object, 801
- Substation Display, 473, 838
- Substation Display Options, 838
- Substation Field Options, 295
- Substation Fields on Onelines, 712
- Substation Information, 293, 391
- Substation Records, 232
- Substation View, 148, 473
- Substations, 293, 391, 473, 711, 789
- Super Area, 226, 439, 442, 705
- Super Area Display Objects, 698
- Super Area Display Options Dialog, 699
- Super Area Information, 439, 442
- Supplemental Classification, 847
- Supplemental Data, 847
 - Creating Records, 847
 - Fields, 758
 - Linking Display Objects, 848
- Switched Shunt, 262, 380, 383, 415, 733, 839, 1179
- Switched Shunt Fields, 383, 734, 809
- Switched Shunt Information, 380, 383, 415, 1179
- Switched Shunts, 214, 219, 380, 383, 415, 420, 474, 734, 787, 963, 964, 1179
- System Scaling, 924

T

- Tap, 255, 965
- Tapping Transmission Lines, 965
- Taps, 966
- Terms, 871
 - x-y, 871
- Text, 137, 174, 754, 755, 809, 810, 827
- Text Fields, 754
- Text/font, 806
- Three Winding Transformers, 257, 378, 729
- Throw Over, 1050
- Thumbnail, 826, 834
 - Background, 835
 - Size, 835
- Thumbnail View, 826, 834
- Tie Lines, 434
- Tie Lines between Areas, 223
- Tie Lines between Zones Display, 225
- Time Step Simulation, 1114, 1115, 1117, 1120, 1121, 1122, 1124, 1125, 1126, 1127, 1131, 1132, 1133, 1134, 1135, 1139, 1140, 1141, 1142, 1145, 1146, 1147, 1150, 1152, 1154, 1156, 1162, 1166, 1167, 1170
 - Binding Constraint Dialog, 1131
 - Change Timepoint Time Dialog, 1141
 - Custom Results Selection Dialog, 1133
 - Dialog, 1117
 - Input, 1125
 - Insert/Scale Column Dialog, 1143
 - Loading Hourly Input Data, 1145
 - Matrix Grids, 1126
 - New Timepoint Dialog, 1140
 - Options, 1135
 - Pages, 1121
 - Quick Start, 1115
 - Results, 1132
 - Results Grid Pages, 1134
 - Results: Constraints, 1127
 - Running a Timed Simulation, 1166
 - Running OPF and SCOPF, 1167
 - Schedule Dialog, 1147

Schedule Subscription Dialog, 1152
 Schedule Subscriptions Page, 1154
 Schedules Page, 1150
 Setting up Scheduled Input Data, 1146
 Specifying and Maintaining a List of Timepoints, 1139
 Storing Input Data and Results, 1170
 Summary, 1122, 1124
 Toolbar, 1120
 TSB Case Description Page, 1142
 TLR, 990, 991, 994, 996
 TLR Multiple Device Type, 994
 TLR sensitivities, 990, 991
 TLR Sensitivities, 996
 Toggle Full Screen, 825
 Tools, 800, 803
 Top, 41, 805, 810
 level, 810
 Topological Differences, 482, 907, 922
 Topology Processing, 538, 1486, 1488, 1490, 1491, 1492, 1493, 1494, 1495, 1496, 1498, 1500, 1501, 1502, 1506, 1507, 1508, 1509, 1510, 1511, 1513, 1514
 Total Digits, 808
 Transactions, 220, 222, 275, 276, 468, 682
 Transfer, 1361, 1371
 Transfer Limit, 1371
 Transfer Limit For Scenario, 1361
 Transfer Limiters, 1361, 1371
 Transfer Limiters Display, 1371
 Transfer Result Reporting Options, 1348
 Transformer AVR Dialog, 340, 370, 408
 Transformer Control, 337, 365, 1163
 Transformer Fields, 728, 809
 Transformer Impedance Correction, 369
 Transformer Impedance Correction Tables, 414
 Transformer Options, 320
 Transformer Phase Shifting, 374, 412
 Transformers, 214, 246, 253, 255, 257, 283, 320, 325, 337, 340, 348, 365, 368, 369, 370, 372, 374, 376, 378, 404, 408, 410, 412, 414, 689, 719, 726, 728, 729, 915, 963, 964, 1163, 1175
 Control, 337, 365
 Fields, 348, 376
 Transient Stability, 540, 1518, 1519, 1523, 1526, 1527, 1528, 1530, 1532, 1534, 1535, 1538, 1544, 1545, 1547, 1550, 1552, 1555, 1556, 1557, 1562, 1564, 1565, 1567, 1571, 1573, 1577, 1582, 1584, 1586, 1588, 1590, 1592, 1593, 1595, 1597, 1598, 1601, 1603, 1605, 1606, 1608, 1611, 1613
 Block Diagrams, 1544
 Data, 1534, 1535, 1544
 Distributed Computing, 1564
 General, 1556
 Generic Limit Monitors, 1562
 Options, 1555, 1556, 1557, 1559, 1562, 1564
 Power System Model, 1557
 Result Options, 1559
 Results, 1603, 1605
 Results Storage, 1565, 1567, 1571
 Single Machine Infinite Bus, 1611
 SMIB, 1611
 Transient Limit Monitors, 1597, 1598, 1601
 Transmission Line, 354, 355
 DC Actual Flow, 355
 DC Inverter Parameters, 354
 Transmission Line Information, 1175
 Transmission Line Options, 320
 Transmission Line Parameter Calculator, 327
 Transmission Line Transformer Options Display, 323
 Transmission Line/Transformer Options, 324, 337, 342, 365
 Parameters/Display, 324
 Series Capacitor, 342
 Transformer Control, 337, 365
 Transmission Lines, 214, 246, 253, 260, 283, 320, 325, 345, 350, 351, 353, 357, 404, 717, 718, 724, 725, 726, 783, 915, 963, 964, 1175
 DC, 350
 DC Line Parameters, 351
 DC Rectifier Parameters, 353

- Fields, 345
- Transmission Loading Relief, 990, 991, 994, 996
- Transmission Loading Relief Sensitivities, 990, 991, 996
- Turns Ratio, 369
- Tutorial, 19, 1097, 1098, 1669
 - Contingency Analysis - Page 10, 1111
 - Contingency Analysis - Page 11, 1113
 - Contingency Analysis - Page 4, 1103
 - Contingency Analysis - Page 5, 1105
 - Contingency Analysis - Page 6, 1106
 - Contingency Analysis - Page 7, 1108
 - Contingency Analysis - Page 8, 1109
 - Contingency Analysis - Page 9, 1110
- Tutorial Contingency Analysis, 1097, 1669
- Tutorial Creating a New Case, 1618
- Tutorial Solving an OPF, 1662
- Tutorial Starting with an Existing Case, 1647
- Tutorial Contingency Analysis, 1098
- Tutorials, 1097, 1618, 1619, 1621, 1623, 1624, 1627, 1629, 1631, 1633, 1636, 1638, 1642, 1644, 1647, 1648, 1649, 1650, 1651, 1652, 1653, 1654, 1655, 1656, 1657, 1658, 1659, 1660, 1661, 1662, 1663, 1664, 1665, 1666, 1667, 1669

U

- Uncheck, 814
 - Anchored, 814
- Undo, 819
- Unit Type, 240
- Unlinked Display Objects, 844
- Unlinked Objects, 798, 844
 - Highlighting, 798
- Unused Bus Numbers, 952
- Use, 814
 - Display/Size Tab, 814
- Use Absolute Values, 827
 - Mvar, 827
 - MW, 827

- Use Area/Zone/Owner Filters, 33, 138, 142
- Use Background Fill, 809
- Use Conditional Display, 812
- User-Defined Case Information, 111, 118, 210
- Using Difference Flows, 480, 905, 920
- Using the Online Alignment Grid, 799

V

- Validate, 23
- Variable O&M, 239
- View, 53, 470, 688, 743, 826, 834, 874
- View Name, 874
- Violations, 1055
- Visualize, 974, 976
 - Power Flows, 974, 976
 - PTDFs, 974
- Visualizing, 660, 828
 - Out-of-Service Elements, 828
- Voltage Adequacy, 1180
- Voltage Control, 395
- Voltage Gauges, 709, 710
- Voltage Sensitivities, 1000
- Voltage Stability, 1180

W

- Warning, 829
- Warning Color, 829
- Warning Percent, 829
 - Default values, 829
- Warning Size Scalar field, 829
- Web, 15
- Web Publishing, 96, 135
- Weighted Transmission Loading Relief, 995
- Why is this field or line disabled, 129
- Wind Control Mode, 302
- Window, 17, 23, 140
- Windows Basics, 17
- Word, 841
- Word Processor, 841

X

X Grid Spacing, 798

XF, 255

x-y, 871, 874

Y

Y Grid Spacing, 798

Ybus, 97, 285

Z

Zero Sequence, 1176

Zone, 153, 308, 385, 401, 418, 703, 795, 801

Object, 801

Zone Display Objects, 696

Zone Display Options Dialog, 697

Zone Fields, 703

Zone Information, 385, 418

Zones, 153, 215, 264, 420, 474

Zoom, 40, 53, 795, 798, 801, 804, 810, 811, 812, 828, 829, 834, 872

Area, 53

In, 53

Levels, 53, 795, 798, 810, 811, 812, 829, 834, 872, 874

Out, 53

Pan, 872

Percentage, 829

Toolbar, 53

Zoom/Pan/Find, 801, 810, 872