Virtuoso Analog Design Environment GXL User Guide

Product Version 6.1.6
November 2014

Contents

Preface

You can perform design optimization and multi-technology simulation on your designs in the ADE GXL environment. ADE GXL eliminates redundant data entry and simplifies moving between design tasks. You can also customize your user interface (see the *Virtuoso® Design Environment User Guide*).

ADE GXL Product Features

From the ADE GXL environment, you can access all features of ADE XL and the following:

- Set up and run optimizations
- Multi-technology simulation
- Set up and run parasitic resimulation

Licensing in Analog Design Environment (ADE) GXL

For information on licensing in the Virtuoso design environment, see *Virtuoso Software Licensing and Configuration Guide*.

Related Documents for ADE GXL

The following documents provide more information about the topics discussed in this guide.

- *Virtuoso Schematic Editor User Guide* describes Cadence's schematic editor.
- *Virtuoso Analog Design Environment XL User Guide* describes the ADE XL environment.
- *Virtuoso Parasitic Estimation and Analysis User Guide* describes the Cadence parasitic simulation product.
- *Spectre Circuit Simulator User Guide* and *Spectre Circuit Simulator Reference* describe Cadence's Spectre analog circuit simulator.

- *SpectreRF Simulation Option User Guide* describes Cadence's RF circuit simulation option.
- *UltraSim User Guide* describes Cadence's multi-purpose single engine, hierarchical simulator, designed for the verification of analog, mixed signal, memory, and digital circuits.
- *Virtuoso AMS Designer Simulator User Guide* describes Cadence's AMS mixed-signal circuit simulator.
- *Virtuoso Visualization and Analysis Tool User Guide* contains product information for waveform viewing and post-processing of simulation results.
- *Component Description Format User Guide* describes Cadence's Component Description Format (CDF) for describing parameters and the attributes of parameters of individual components and libraries of components.
- *Analog Expression Language Reference* contains concept and reference information about the Analog Expression Language (AEL).

Additional Learning Resources

Cadence provides various Rapid Adoption Kits that you can use to learn how to employ Virtuoso applications in your design flows. These kits contain workshop databases, designs, and instructions to run the design flow.

Cadence offers the following training course on the Virtuoso Analog Design Environment GXL flow:

- Virtuoso Analog Design Environment
- Virtuoso Schematic Editor
- Analog Modeling with Verilog-A
- Behavioral Modeling with Verilog-AMS
- Real Modeling with Verilog-AMS
- Spectre Simulations Using Virtuoso ADE
- Virtuoso UltraSim Full-Chip Simulator
- Virtuoso Simulation for Advanced Nodes
- Virtuoso Electrically-Aware Design with Layout Dependent Effects

For further information on the training courses available in your region, visit the Cadence Training portal. You can also write to training_enroll@cadence.com.

Note: The links in this section open in a new browser. The course links initially display the requested training information for North America, but if required, you can navigate to the courses available in other regions.

Third Party Tools for ADE GXL

To view any `.swf` multimedia files, you need:

- Access to the Cadence Online Support website.
- Flash-enabled web browser, for example, Internet Explorer 5.0 or later, Netscape 6.0 or later, or Mozilla Firefox 1.6 or later. Alternatively, you can download Flash Player (version 6.0 or later) directly from the Adobe website.
- Speakers and a sound card installed on your computer for videos with audio.

Typographic and Syntax Conventions

The following typographic and syntax conventions are used in this manual.

`text`	Indicates text you must type exactly as it is presented.
`z_argument`	Indicates text that you must replace with an appropriate argument. The prefix (in this case, *`z_`*) indicates the data type the argument can accept. Do not type the data type or underscore.
`[ ]`	Denotes an optional argument. When used with vertical bars, they enclose a list of choices from which you can choose one.
`{ }`	Used with vertical bars, they denote a list of choices from which you must choose one.
|	Separates a choice of options.
…	Indicates that you can repeat the previous argument.
=>	Precedes the values returned by a Cadence® SKILL language function.

`/`	Separates the possible values that can be returned by a Cadence SKILL language function.
text	Indicates names of manuals, menu commands, form buttons, and form fields.

1

Getting Started in the Environment

When you launch the environment, *ADE GXL* and *Parasitics* appear on the menu banner.

The ADE GXL environment consists of a set of toolbars and assistant panes that make up your workspace. You can load a Cadence workspace or create and load a custom workspace. You can specify what workspace to load for a given cellview. For more information about workspaces, see "Getting Started with Workspaces" in the *Virtuoso Design Environment User Guide*.

The *Parasitics* menu lets you access Virtuoso® Parasitic Estimation and Analysis to investigate the effects of parasitics on your circuits. You can report on parasitics that exist in your design, show or hide them on your design, and create refined extracted cell views. For more information, see the *Virtuoso Parasitic Aware Design User Guide*.

See the following topics for more information about the ADE GXL environment:

- Launching the ADE GXL Environment on page 12
- Using the ADE GXL Banner Menu on page 12
- "Specifying the Run Mode" in the *Virtuoso® ADE XL User Guide*

See also

- Chapter 4, "Circuit Optimization" for information about circuit optimization
- Chapter 7, "Multi-Technology Simulation" for information about using multi-technology simulation
- The *Virtuoso® Parasitic Estimation and Analysis User Guide* for information about parasitic resimulation

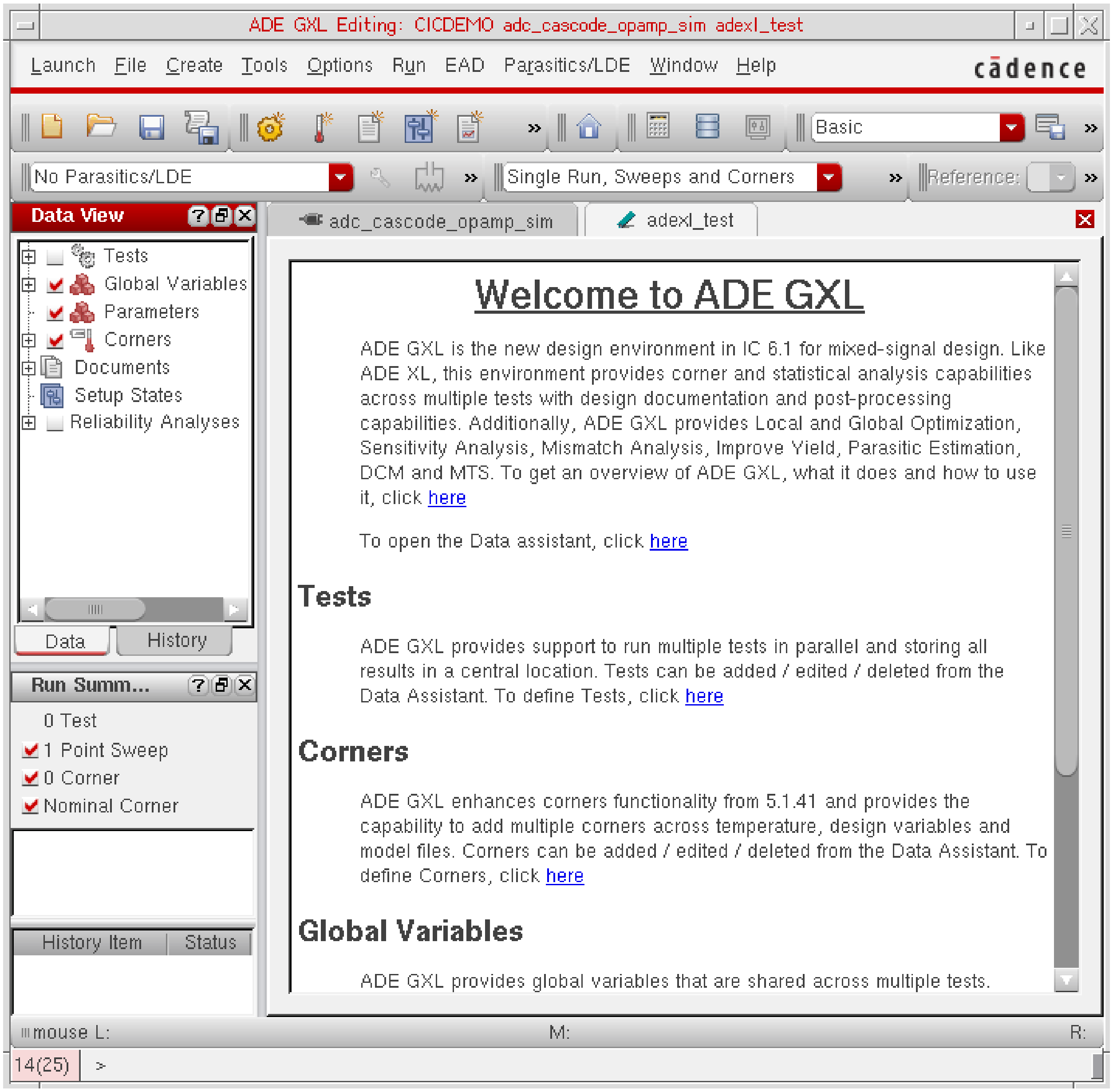
Launching the ADE GXL Environment

To start the environment, do one of the following:

To open the environment from a schematic editing window	➤ In the schematic editing window, choose *Launch – ADE GXL*. The *ADE GXL* environment appears. **Note:** If you have descended into a design hierarchy, the environment returns you to the top level of your design when you choose *Launch – ADE GXL*.
To open an existing ADE GXL cellview from the CIW	**1.** In the CIW, choose *File – Open*. The Open File form appears. **2.** Select an ADE GXL design cellview. The *ADE GXL* environment appears.

Using the ADE GXL Banner Menu

When you launch the ADE GXL environment, *ADE GXL* appears on the menu banner.



The menu entries available are the same as <u>those for ADE XL</u>.

2

Sensitivity Analysis

Sensitivity analysis helps in finding out the sensitivity of specifications to variables, device parameters, statistical parameters and DC operating point parameters. This chapter provides an overview of sensitivity analysis and describes how to run this analysis and view results. It also explains how to create worst case corners based on the results of sensitivity analysis.

For more details, refer to the following sections:

Generating Sensitivity Data

You can generate and view data on the sensitivity of specifications to variables, device parameters, statistical parameters and DC operating point parameters.

Variation of Design Variables and Device Parameters

ADE GXL varies each variable or parameter one-factor-at-a-time (OFAT). To calculate sensitivity, the tool perturbs the variable or parameter value around its nominal value, while leaving other variables as fixed.

For design and PVT variation, ADE GXL provides the following methods:

- OFAT 3-level: In this method, you can specify three values for a variable. For example, 1.9u, 2.0u, 2.1u with nominal value equal to 2.0u. If, instead of three points, you specify a range of values, the tool chooses the following values of the variable for simulation:
  - one step below the nominal value
  - nominal value
  - one step above the nominal value

  For example, if the variable range is 1u:0.1u:5u and the nominal value is 3.6u, the three values used by the tool are: 3.5u 3.6u, and 3.7u.
- OFAT Sweep: In this method, you can specify more than three values. For example, if you specify the variable range as 200n:50n:400n, and the nominal value as 250n, the variable values used for simulation are: 200n, 250n, 300n, 350n, and 400n.
- Hammersley: Select this method to capture both nonlinear relationships and interactions among the design variables. The OFAT methods vary one factor at a time, where as Hammersley generates samples varying more than one variable at a time. Post-process the Hammersley sampling results with Variance Contribution analysis.

  Note: Statistical variables cannot be varied with Hammersley sampling. Run Monte Carlo analysis for statistical analysis.

Variation of Statistical Parameters

For statistical variation, ADE GXL varies each statistical parameter one-factor-at-a-time at 3 levels (values). When the statistical parameter is modeled with normal or log normal distribution, ADE GXL uses the following values for simulation:

- N sigma below the mean value

- mean value
- N sigma above the mean value

When the statistical parameter is modeled with uniform distribution, ADE GXL chooses values based on the percentage of range.

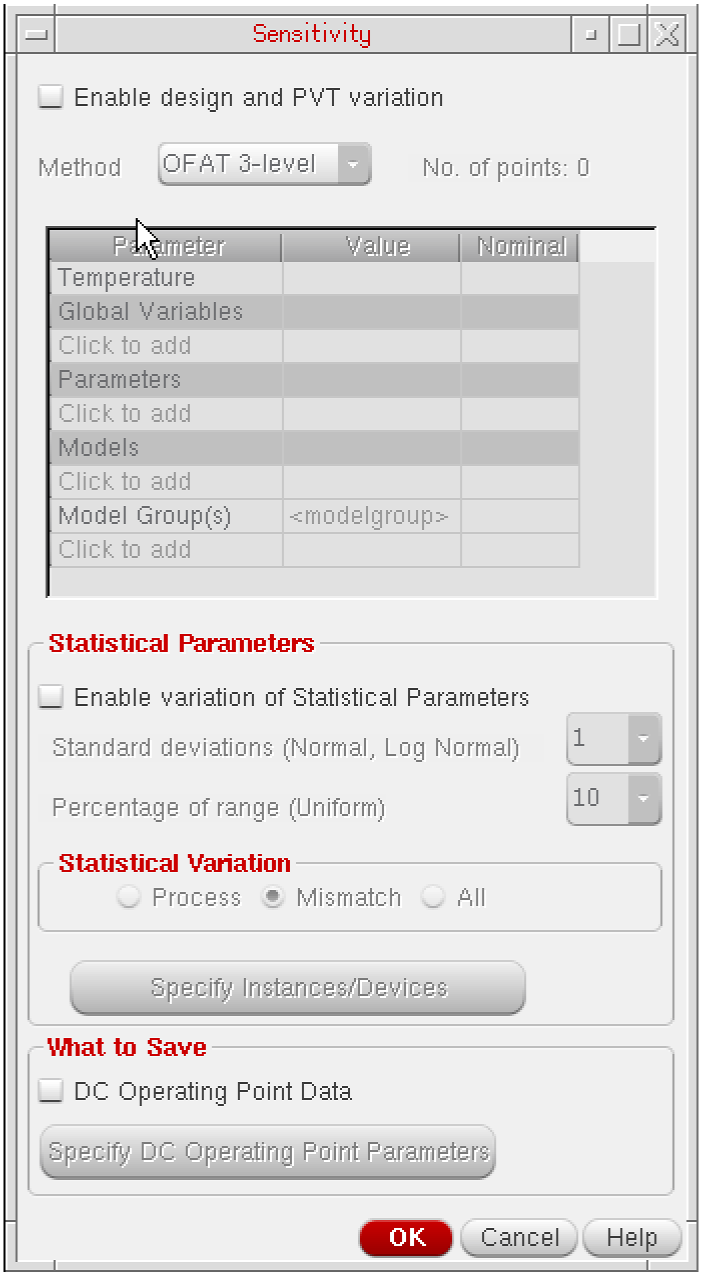
Sometimes the variable whose sensitivity is being determined is at its boundary of possible values (as specified by the variable range and the nominal value). In this case, ADE GXL cannot determine a higher value (if the current value is at the upper boundary) or lower value (if at lower boundary) for computing its sensitivity. In this case, ADE GXL only uses two points for calculating the sensitivity and the correlation will always be 1 or -1, since two points will always form a straight line. However, the regression values (for this variable) are still applicable (since they represent the slope of the line).

For boundary conditions where there are only two points available for simulation, the correlation is +1 or -1, depending upon whether the regression coefficient is positive or negative. However, it does not reflect the true meaning of correlation because you need to specify at least three points for calculating the correlation coefficients.

To generate and view the sensitivity data:

1. Select *Sensitivity Analysis* in the *Select a Run Mode* drop-down list on the Run toolbar, then click the *Simulation Options* button on the Run toolbar.

   The Sensitivity form appears, as shown below.



Note: The *Edit Reference Point* command on toolbar is disabled when you select *Sensitivity Analysis* in the *Select a Run Mode* drop-down list.

2. Select the *Enable Design and PVT Variation* check box if you want to vary temperature, global variables, device parameters, or model files for sensitivity analysis.

   Ensure that if you select this check box, you have specified at least one variable or parameter in the Variables and Parameters assistant. The variables or parameters that you have included in this form appear with a strike through in the Variables and Parameters assistant and the Data View pane.

Also ensure that you disable all other variables or parameters that have sweep values in the Variables and Parameters assistant and are not overridden in the Sensitivity form. This is because Sensitivity Analysis does not consider variables with sweep values in the Variables and Parameters assistant and displays an error.

Then, do the following:

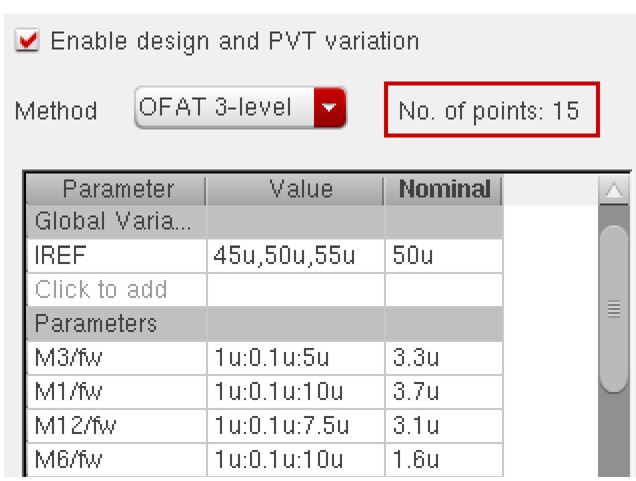
a. In the *Method* field, specify the method to be used to vary global variable and parameter values. You can select any one of the following two values:

- `OFAT 3-level`
- `OFAT sweep`
- `Hammersley`

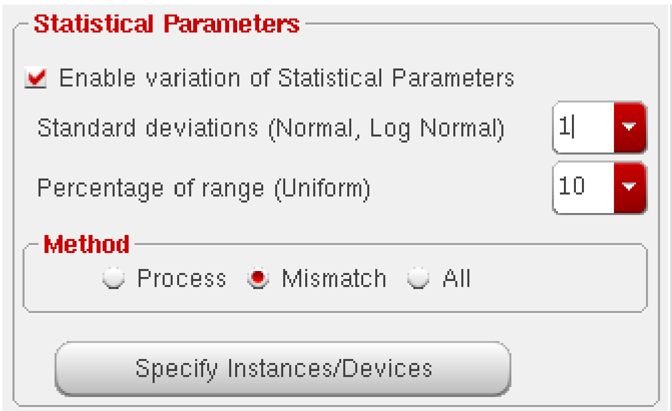
For more details on these methods, refer to Variation of Design Variables and Device Parameters.

b. In the table given below the *Method* field, select the design variables, parameter, or model files that you want to vary. For more details, refer to Specifying Variables, Parameters, and Model Files to be Varied for Sensitivity Analysis on page 23.

c. Depending on the range of values and nominal value and the value in the *Method* list, the tool calculates the total number of data points for the sensitivity analysis and displays the count in the *No. of Points* label to the right of the *Method* field, as shown below:



3. Select the *Enable variation of Statistical Parameters* check box if you want to vary statistical process and mismatch parameters.



Then, do the following:

a. In the *Standard deviations (Normal, Log Normal)* field, specify the number of standard deviations for statistical parameters with normal or log normal distribution.

b. In the *Percentage of Range (Uniform)* field, specify the percentage range by which statistical parameters with uniform distribution need to be varied.

Note: The value specified must be a number greater than `0` and no more than `50`.

c. In the *Method* group box, select one of the following statistical variations:

Process	for process statistical variations
Mismatch	for per-instance statistical variations
All	for both process and per-instance statistical variations

d. (Optional) In the step c above, if you select *Mismatch* or *All*, by default, the tool varies mismatch statistical parameters for all devices and instances in the design. Click *Specify Instances/Devices* to select specific instances and devices for which you want to analyze the impact of statistical variations. For more details, refer to Selecting Instances and Parameters for Mismatch Variation on page 28.

4. (Optional) By default, you cannot view the sensitivity data for DC operating point parameters because the data is not saved in the results database. To save and view sensitivity data for specific DC operating point parameters, do the following:

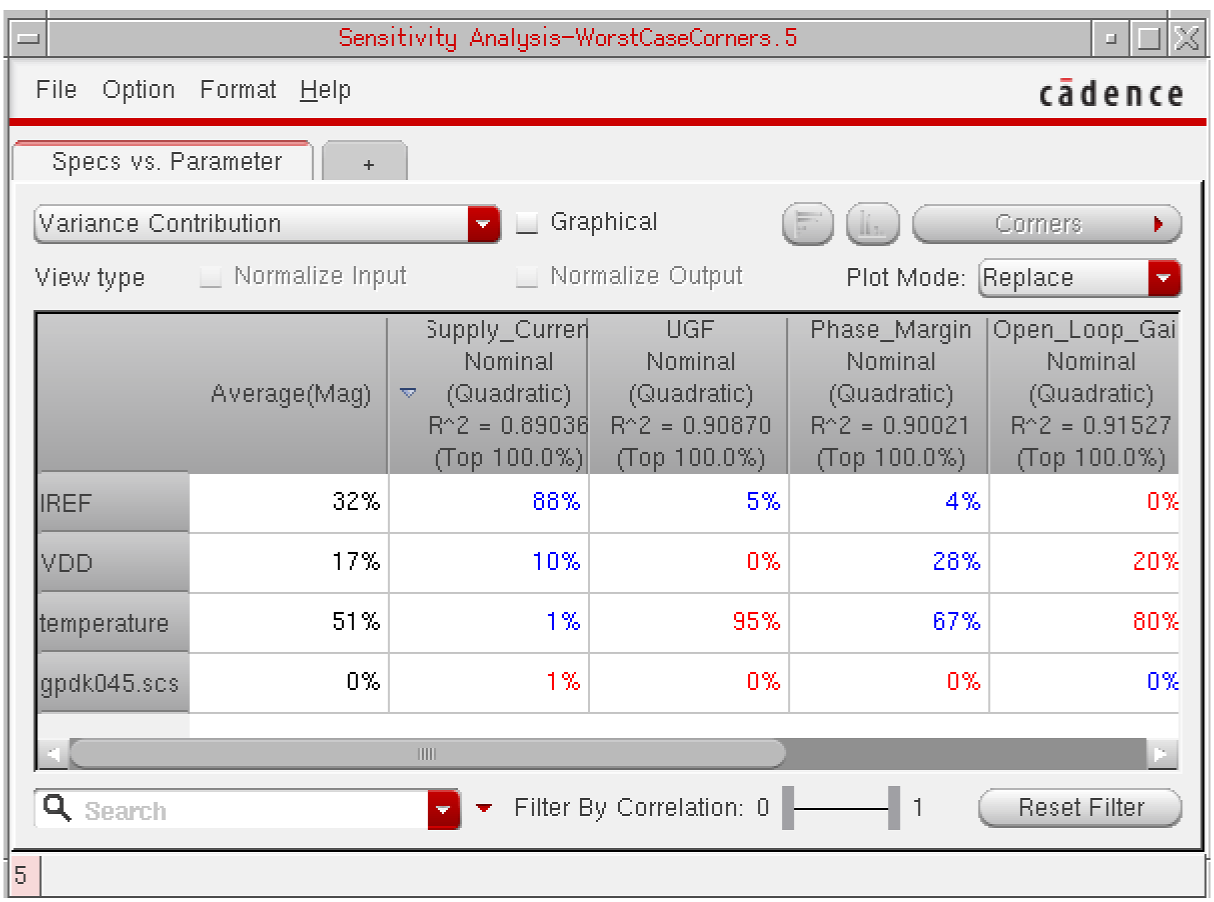
a. Select the *DC Operating Point Data* check box.

b. Click the *Specify DC Operating Point Parameters* button to specify the DC operating point parameters for which you want to save and view sensitivity data. For more information, see Saving Sensitivity Data for DC Operating Point Parameters on page 34.

5. Click *OK* to save the changes and close the Sensitivity form.

6. Click the *Run Simulation* button on the Run toolbar.

After the run is complete, ADE GXL displays the simulation results on the Results tab. In addition, it also opens the **Sensitivity Analysis** window that displays the sensitivity data for different specs or parameters. By default, this window displays the Variance Contribution results in percentage format, as shown below.

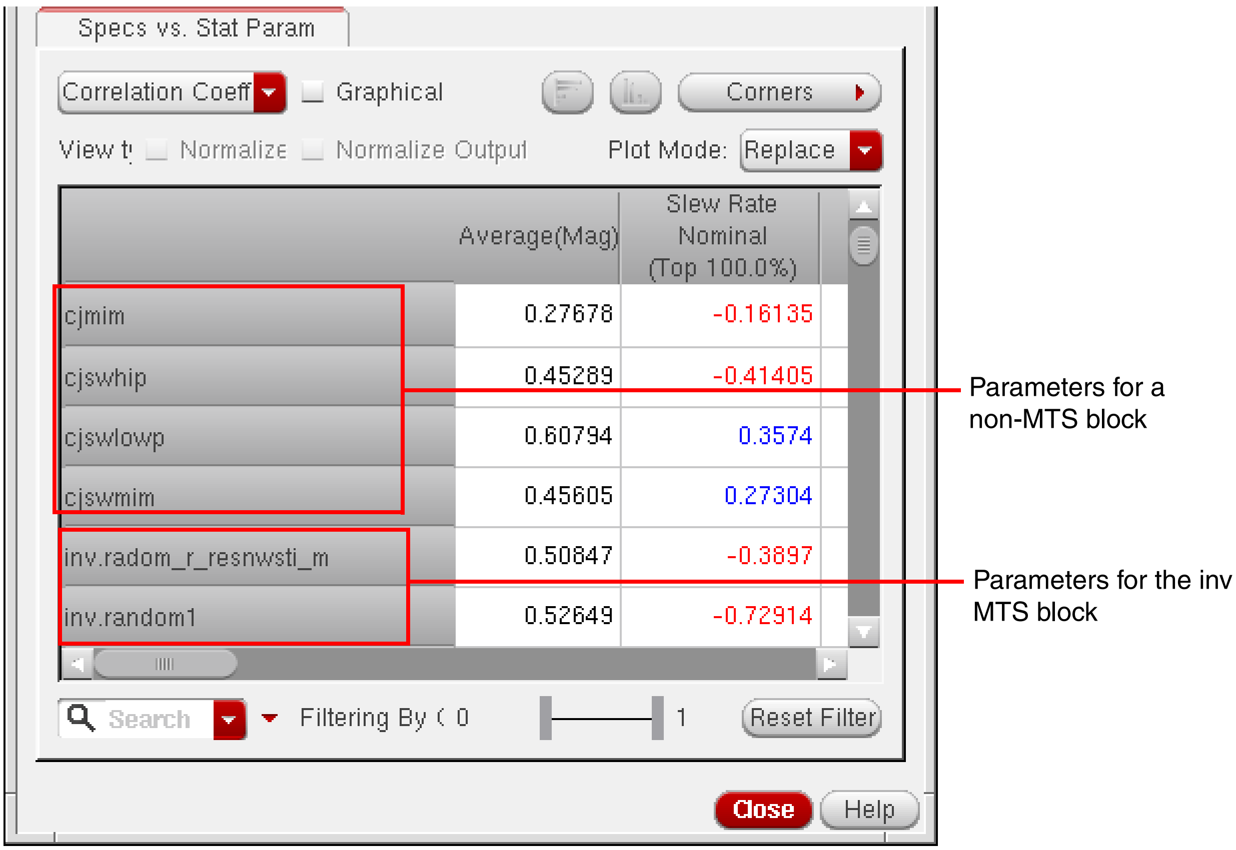


You can view the data in other formats. For more details, refer to Working with Sensitivity Data.

Support for Multi-Technology Simulations in Sensitivity Analysis

If you have enabled multi-technology simulations for a test, in the Sensitivity Analysis results window, the parameter names for MTS blocks are prefixed with the name of their corresponding block.

For example, in the sensitivity analysis results displayed below, the names of parameters for the `inv` MTS block are prefixed with `inv`.



Troubleshooting a Design Point from the Sensitivity Results

To troubleshoot a point from the sensitivity results, in the Results tab, right-click in the data cell and choose *Troubleshoot Point*.

Note: If you troubleshoot a design point from existing sensitivity analysis results, the Sensitivity Analysis window is not displayed after the run. This is because sensitivity analysis cannot be performed on the results of a troubleshoot point. The following commands are also disabled for the sensitivity analysis results of a troubleshoot point:

- *Sensitivity Analysis* command in the shortcut menu of a history item on the *History* tab

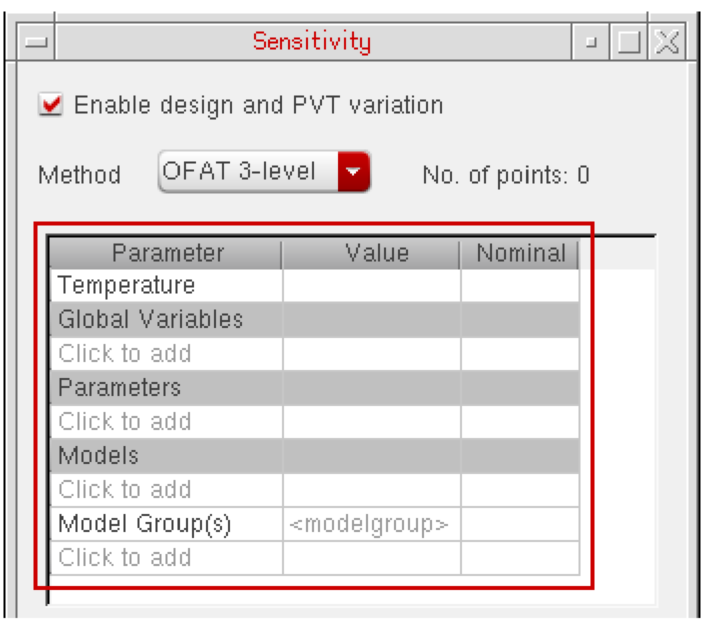
-  button on the toolbar in the *Results* tab

See the following topics for more information about using the Sensitivity form:

- Specifying Variables, Parameters, and Model Files to be Varied for Sensitivity Analysis on page 23
- Selecting Instances and Parameters for Mismatch Variation on page 28
- Viewing Sensitivity Data on page 48
- Working with Sensitivity Data on page 53
- Plotting Sensitivity Analysis Results on page 76
- Highlighting Associated Devices in the Schematic on page 73
- Saving the Sensitivity Data on page 91

Specifying Variables, Parameters, and Model Files to be Varied for Sensitivity Analysis

You can specify the design variables, parameters, or model files that you want to vary for sensitivity analysis in the table given on the Sensitivity form.



In this table, you can select the name of a variable and specify values for which you want to vary the variable and a nominal value for it.

To vary temperature:

1. Double-click in the *Value* cell for `Temperature` and specify a range or a set of values for which you want to vary temperature.
2. Double-click in the *Nominal* cell for temperature and open the drop-down list.

   All the values specified in the *Value* cell are displayed in the list.
3. Select a nominal value from this list.

   Alternatively, you can get the value of temperature from the test or a reference point. For this, right-click in the *Nominal* cell for `Temperature` and choose an appropriate command from the following options:

   - *Get Value from Test*: Copies the value of temperature from the test.
   - *Get Value from Reference Point*: Copies the value of temperature from the reference point created using the Edit Reference Point form.

   Note: If the value taken from the test or reference point does not exist in the *Value* cell, the tool automatically adds the nominal value to the existing values. For example, if you specify -40 and 85 as two values for temperature and get the nominal value from test, which is 27, the tool adds 27 to the value list.

Parameter	Value	Nominal
Temperature	-40 27 85	27

To select global variables to be varied:

1. Click on *Click to add* in the Global Variables section in the table.

   A list of global variables given in the Data View pane is displayed in the cell.
2. From the list, select a global variable that you want to vary.

   The tool gets the sweep values and the nominal value specified for the variable in the Variables and Parameters assistant and displays it in the *Value* and *Nominal* cells, respectively, as shown below.

Parameter	Value	Nominal
Temperature		
Global Variables		
IREF	45u,50u,55u	50u
Click to add		
Parameters		
Click to add		

If required, you can change the range or nominal value for a variable in this table. For more details, refer to Changing the Parameter Value and Changing the Nominal Value.

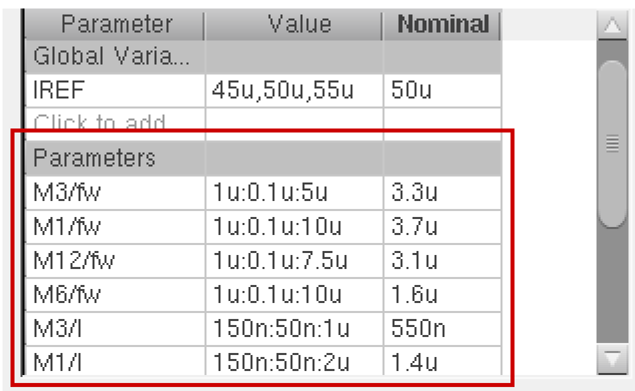
To select design parameters to be varied:

1. Click on *Click to add* in the Parameters section in the table.

   A list of parameters already defined in the Variables and Parameters assistant is displayed in the cell.

2. From the list, select the parameter that you want to vary.

   The tool reads the range of values and design value for the selected parameter from the Variables and Parameters assistant and displays it in the *Value* and the *Nominal* column, respectively, as shown below.



Parameter	Value	Nominal
Global Varia...		
IREF	45u,50u,55u	50u
Click to add		
Parameters		
M3/fw	1u:0.1u:5u	3.3u
M1/fw	1u:0.1u:10u	3.7u
M12/fw	1u:0.1u:7.5u	3.1u
M6/fw	1u:0.1u:10u	1.6u
M3/l	150n:50n:1u	550n
M1/l	150n:50n:2u	1.4u

   If required, you can change the range or nominal value for a variable in this table. For more details, refer to Changing the Parameter Value and Changing the Nominal Value.

Tip

You can move the mouse over the row of a parameter to view the library/cell/view information for the device parameter.

To select model files to be varied:

1. Click on *Click to add* in the Models section in the table.

   The Add/Edit Model Files form is displayed.

2. Browse and select a model file for which you want to vary different sections.

   The name of the file appears in the cell.

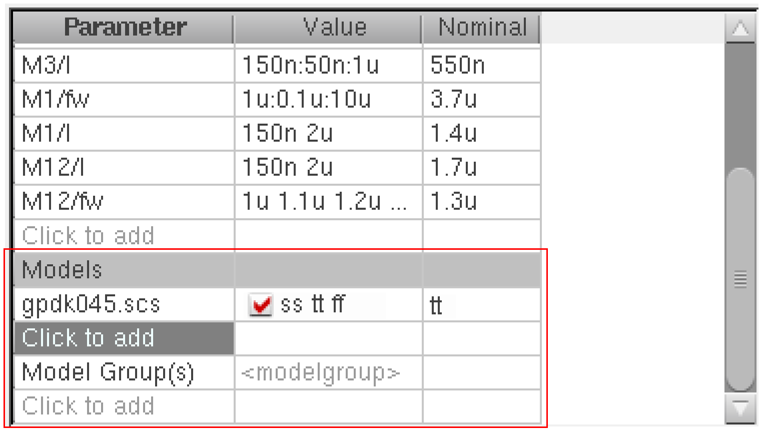
3. Double-click in the *Value* cell.

A list of sections defined in the specified model file appears.

4. Select a section that you want to add to the list.

   You can select multiple sections that you want to vary for the model file.

5. Double-click in the *Nominal* cell and select name of a section that you want to use as a nominal value for the selected model file, as shown below.



Parameter	Value	Nominal
M3/l	150n:50n:1u	550n
M1/fw	1u:0.1u:10u	3.7u
M1/l	150n 2u	1.4u
M12/l	150n 2u	1.7u
M12/fw	1u 1.1u 1.2u ...	1.3u
Click to add		
Models		
gpdk045.scs	ss tt ff	tt
Click to add		
Model Group(s)	<modelgroup>	
Click to add		

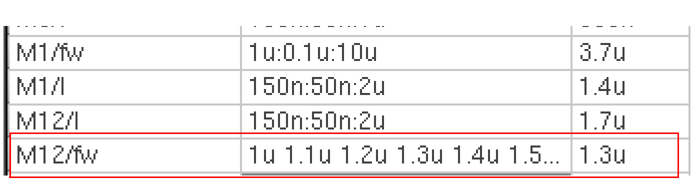
Similarly, you can select model groups that you want to vary for the simulation.

Important

You must define your models so that they respond to the statistical variations you choose. For a Spectre circuit simulator example on how to define your models, see Specifying Parameter Distributions Using Statistics Blocks in the *Virtuoso Spectre Circuit Simulator User Guide*.

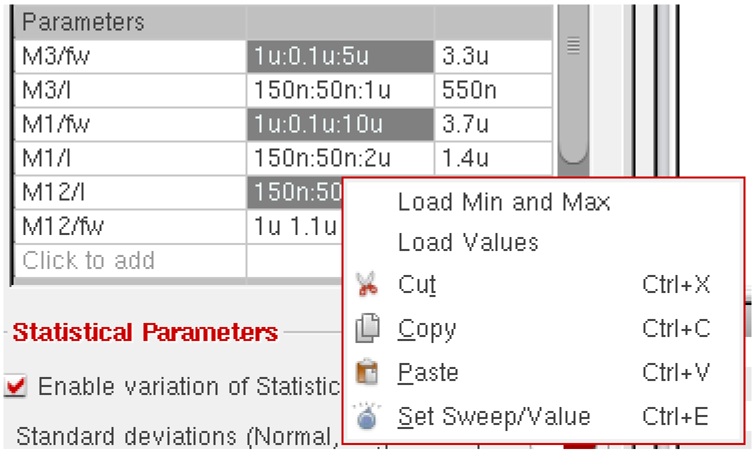
Changing the Parameter Value

To change the values for a parameter, double-click in the *Value* cell for the variable or parameter and edit the value. You can either specify a range or specify a list of space-separated values, as shown below.



M1/fw	1u:0.1u:10u	3.7u
M1/l	150n:50n:2u	1.4u
M12/l	150n:50n:2u	1.7u
M12/fw	1u 1.1u 1.2u 1.3u 1.4u 1.5...	1.3u

Alternatively, you can select one or more values and right-click to display the shortcut menu:



From this menu, choose:

- *Load Values* to load the values defined in the Data View or the Variables and Parameters assistant
- *Load Min and Max* to load only the minimum and maximum values defined in the Data View or the Variables and Parameters assistant.
- *Set Sweep/Value* to edit the exclusion/inclusion list.

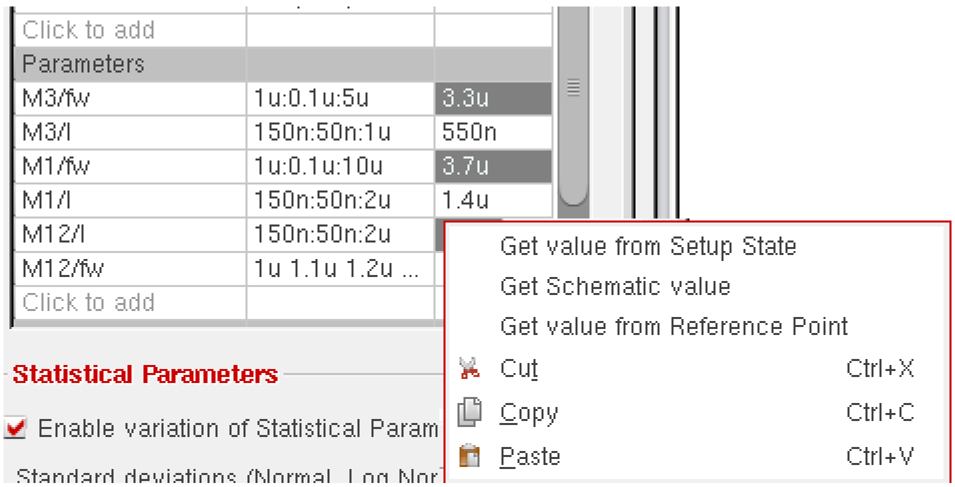
Changing the Nominal Value

The default nominal value is taken from the values of the parameter in the design. To change the nominal value for a variable or parameter, double-click in the *Nominal* cell for the variable or parameter. A list of possible nominal values is displayed. You can either select a value from the list or edit the value. Note that:

- If the variation method is `OFAT 3-level`, a list of all possible nominal values calculated using the range of values is displayed.
- If the variation method is `OFAT Sweep`, all sweep values appear in the list. You can select any value from this or type a different value.

  Note: If you specify a nominal value outside the specified range or the sweep set, the tool does not accept the value and makes the cell blank. A valid nominal value for a parameter is any value from within the range or sweep set corresponding to the parameter.

Alternatively, select one or more cells in the Nominal column and right-click to display the following shortcut menu:



and choose any one of the following commands:

- *Get value from Setup State* to get the nominal value from a saved setup state.

  The Get Nominal Values from Setup State form is displayed. In the *State Name* list, select the name of a saved state. In the *What to Load* section, select *Parameters*. This option copies the values of all the parameters from the selected saved state.

- *Get Schematic value* to get the nominal value from the schematic.

  Note: This command is not available for global variables. Instead, the *Get value from Test* command is available. You can use this command to use the value of variable from the test.

- *Get value from Reference Point* to get the nominal value from the reference table. For this, ensure that you have specified reference values for parameters in the Edit Reference Point form.

  Note: If the value taken from the setup state, schematic, or reference point does not fall in the given value range or sweep set, the tool automatically expands the range or adds the schematic value to the sweep set.

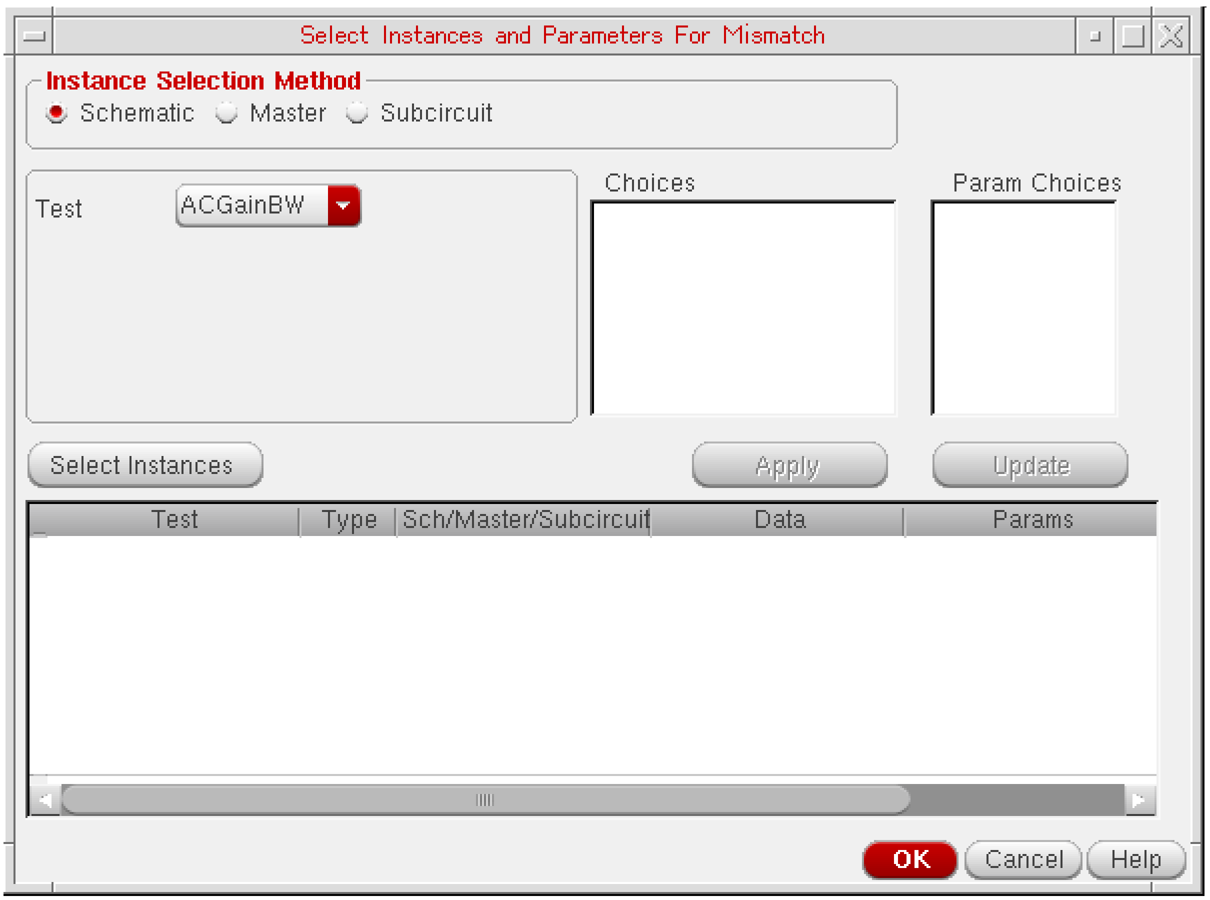
Selecting Instances and Parameters for Mismatch Variation

By default, statistical analysis analyzes mismatch variations for all devices and instances in the design. If the number of instances is large, time to run simulation is very high. You can limit mismatch variation analysis to run for a selected set of instances or devices. For this, click *Specify instances/Devices* on the Sensitivity options form.

The tool netlists the design for the enabled tests and extracts data on the instances modeled by statistical mismatch.

Note: Netlist is created only when you are selecting instances for a set of tests for the first time. When you click on *Specify instances/Devices* for consecutive selections, netlist is created for only those designs/tests that were changed after the previous selections.

After the dummy simulations are run successfully, the Select Instances and Parameters for Mismatch form is displayed, as shown in the following figure.



Only those instances that are modeled with mismatch variation will be listed in the *Choices* lists.

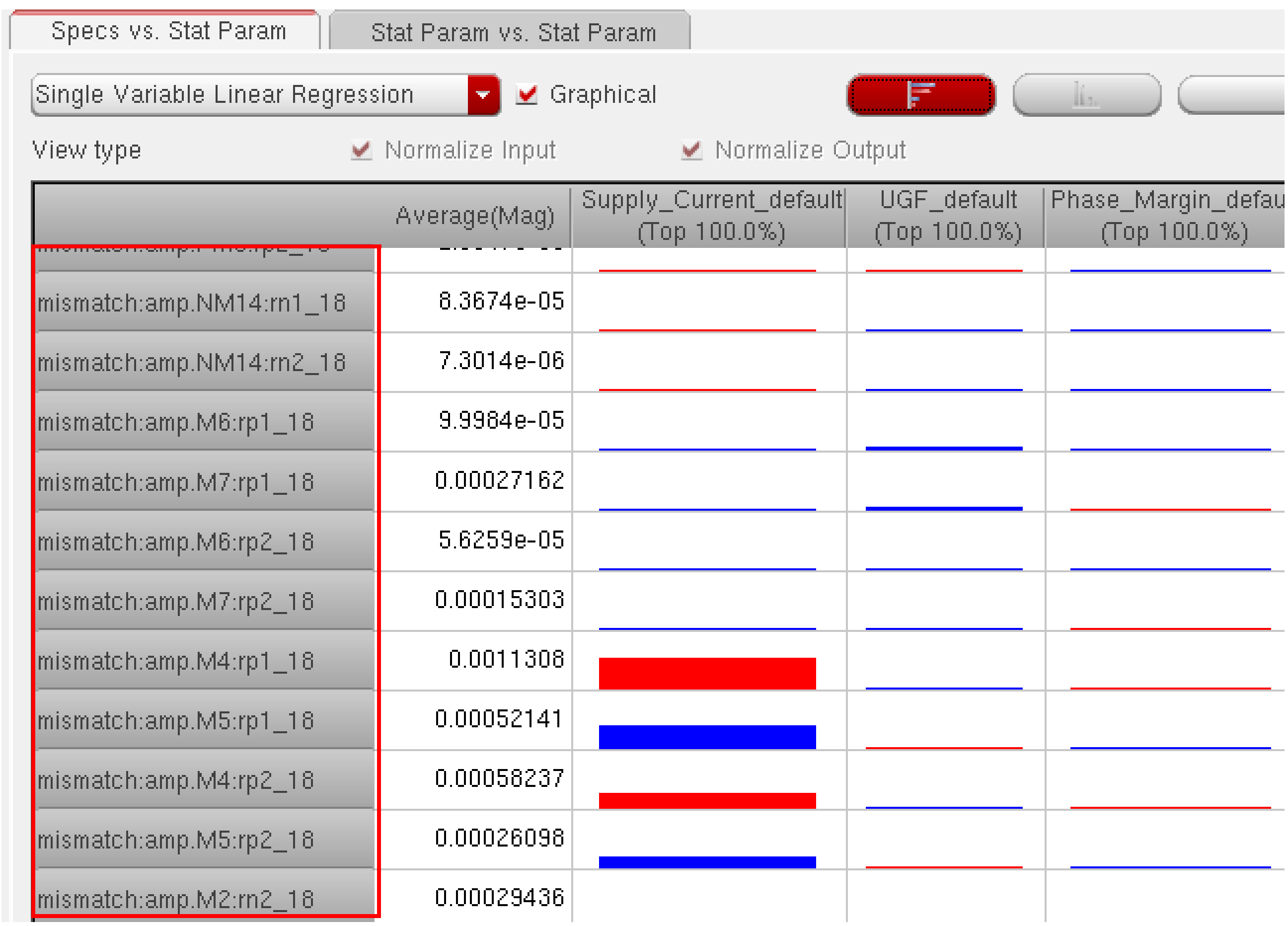
To select instances and parameters, do the following:

1. In the *Instance Selection Method* section, select a method by which you want to select instances. You can select instances in any of the following three ways:

- ❑ By selecting instances on schematic. For this, select the *Schematic* option in the *Instance Selection Method* section. For more details, refer to Selecting Instances from Schematic on page 31.
- ❑ By selecting instances of master cellviews. For this, select the *Master* option in the *Instance Selection Method* section. For more details, refer to Selecting Instances from Master Cellviews on page 32.
- ❑ By selecting subcircuit instances. For this, select the *Subcircuits* option in the *Instance Selection Method* section. For more details, refer to Selecting Instances from Subcircuits on page 33.

2. Click *OK* to save the changes and return to the Sensitivity options form.

Next time when you run sensitivity analysis, the mismatch variation is run for only the selected instances. For example, in the following figure, the variation is limited to only selected instances and parameters.



Selecting Instances from Schematic

To select instances from schematic, do the following:

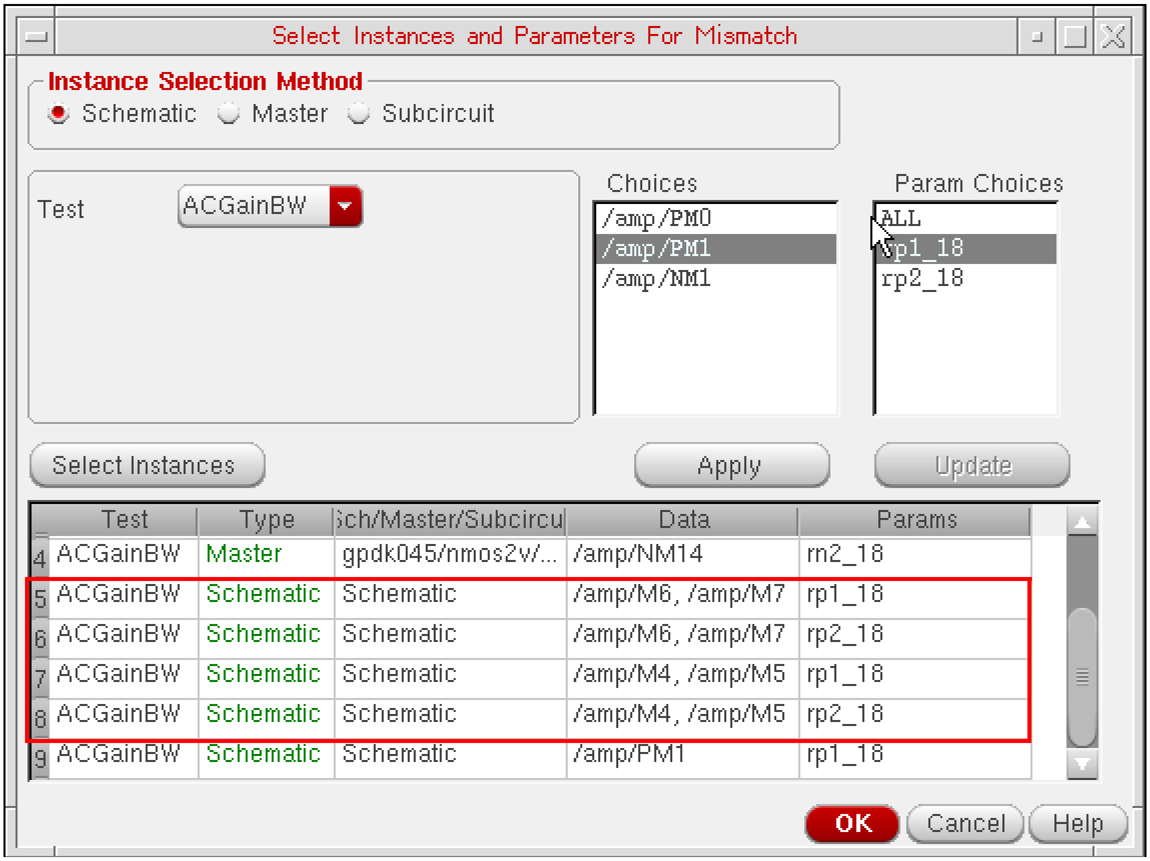
1. In the *Instance Selection Method* section, select the *Schematic* option.
2. In the *Test* list, select the name of test for which you want to open the schematic view.

   Note: Select *All Tests* if you want to select instances for all the tests that are enabled in the *Data View* pane.
3. Click *Select Instances*. The schematic view for the selected test is opened in a new tab.
4. On the schematic tab, hold the *Shift* key and select instances that you want to include for mismatch variation.
5. Press the *Esc* key when done.

   The list of selected instances that are modeled with statistical mismatch variation is displayed in the *Choices* list.

   Note: The selected instances that are not modeled with statistical mismatch variation will not be added to the *Choices* list. Instead, an error message 1692 is displayed with the names of all such instances.
6. In the *Choices* list, select an instance.

   modeled parameters corresponding to the selected instances are displayed in the *Param Choices* list.
7. In the *Param Choices* list, select `ALL` or specific parameters that you want to vary for mismatch variation.
8. Click *Apply* to save the selections as rows in the tabular list, as shown in the following figure.



Selecting Instances from Master Cellviews

To select instances from cellviews, do the following:

1. In the *Instance Selection Method* section, select the *Master* option.

   List of all the libraries, cells, and views are displayed for the selected test.

2. Select the library, cell, and view name for which you want to select instances.

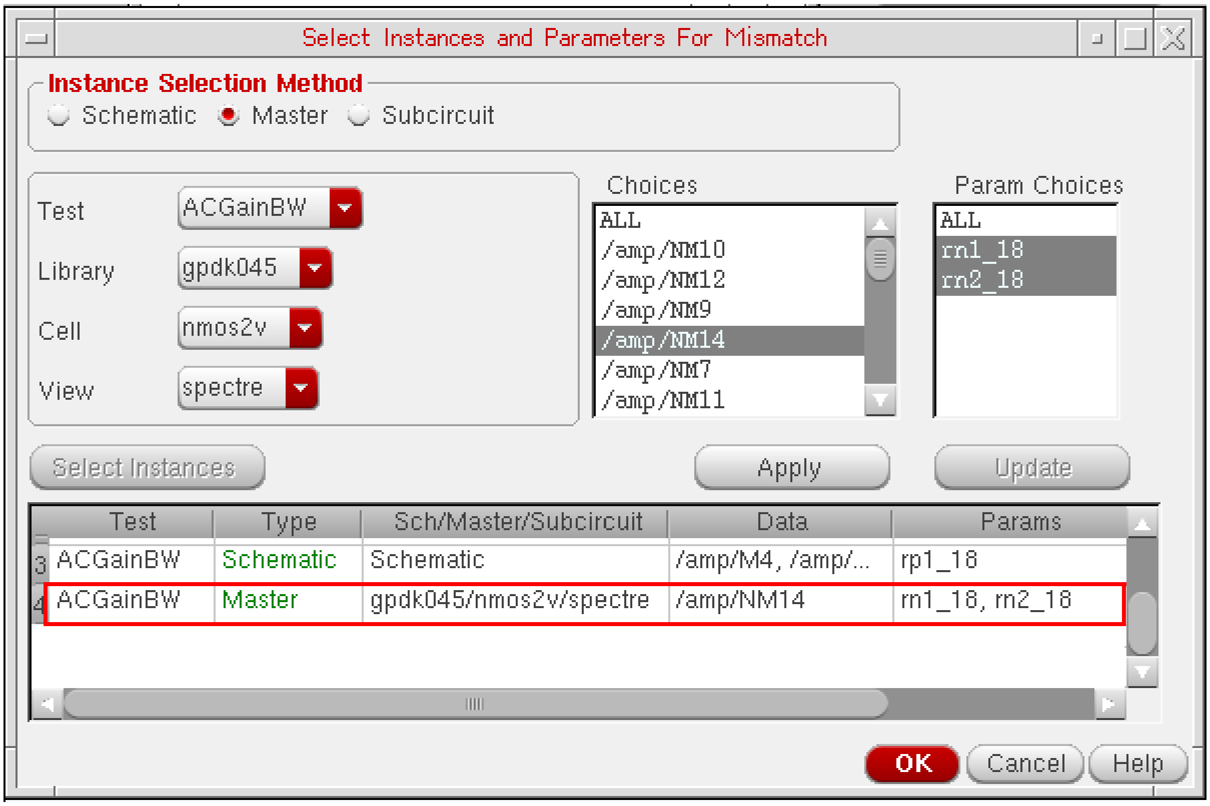
   For the selected cellview, the list of instances that are modeled with statistical mismatch variation are displayed in the *Choices* list.

3. In the *Choices* list, select one or more instances.

   The modeled parameters corresponding to the selected instances are displayed in the *Param Choices* list.

4. In the *Param Choices* list, select `ALL` or specific parameters that you want to vary.

5. Click *Apply* to save the selections as rows in the tabular list, as shown in the following figure.



Selecting Instances from Subcircuits

To select instances from subcircuits, do the following:

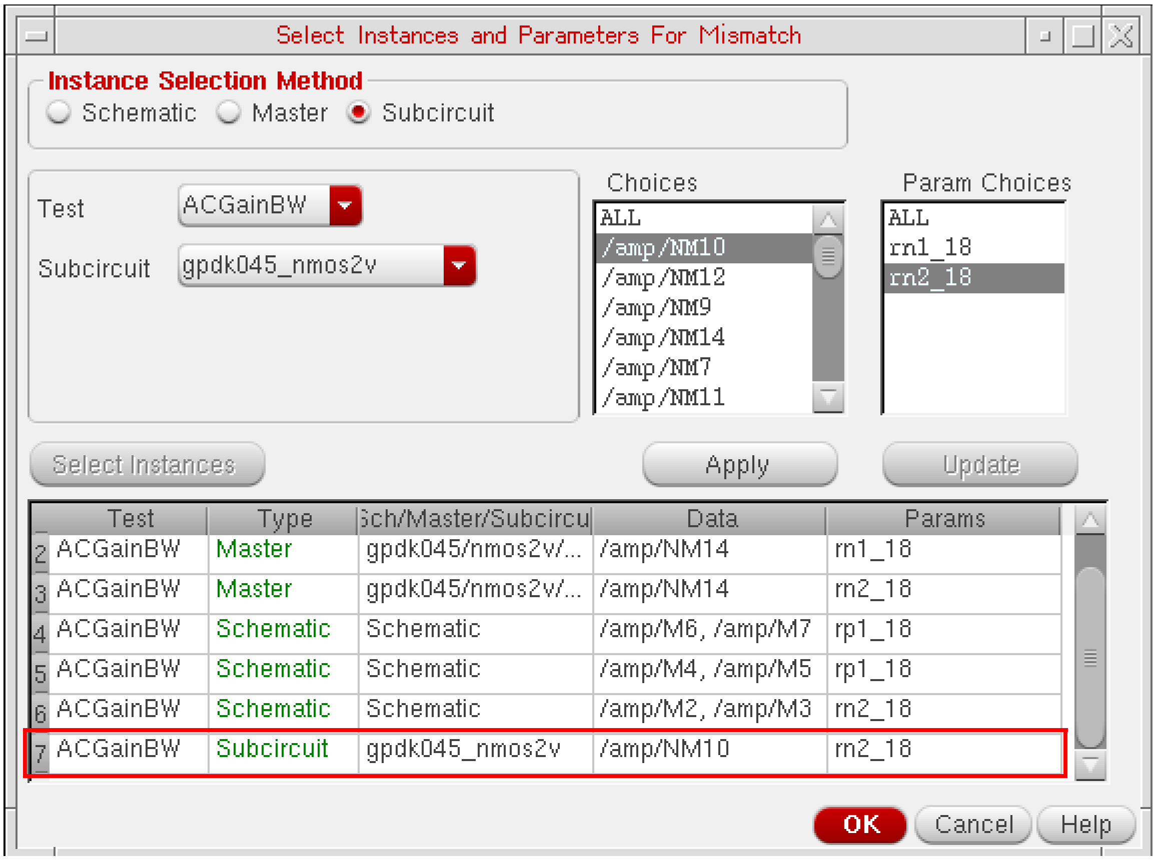
1. In the *Instance Selection Method* section, select *Schematic*.
2. In the *Test* list, select the name of test for which you want to open the schematic view.

   Note: Select *All Tests* if you want to select instances for all the tests that are enabled in the *Data View* pane.
3. In the *Subcircuit* list, choose the subcircuit whose instances you want to select.

   All the instances in the selected subcircuit that are modeled with statistical mismatch variation are displayed in the *Choices* list.
4. In the *Choices* list, select one or more instances.

   The modeled parameters corresponding to the selected instances are displayed in the *Param Choices* list.

5. In the *Param Choices* list, select `ALL` or specific parameters that you want to vary.
6. Click *Apply* to save the selections as rows in the tabular list, as shown in the following figure.



Tip

To delete any selection from the tabular list, select the required row(s), right-click and choose *Delete*.

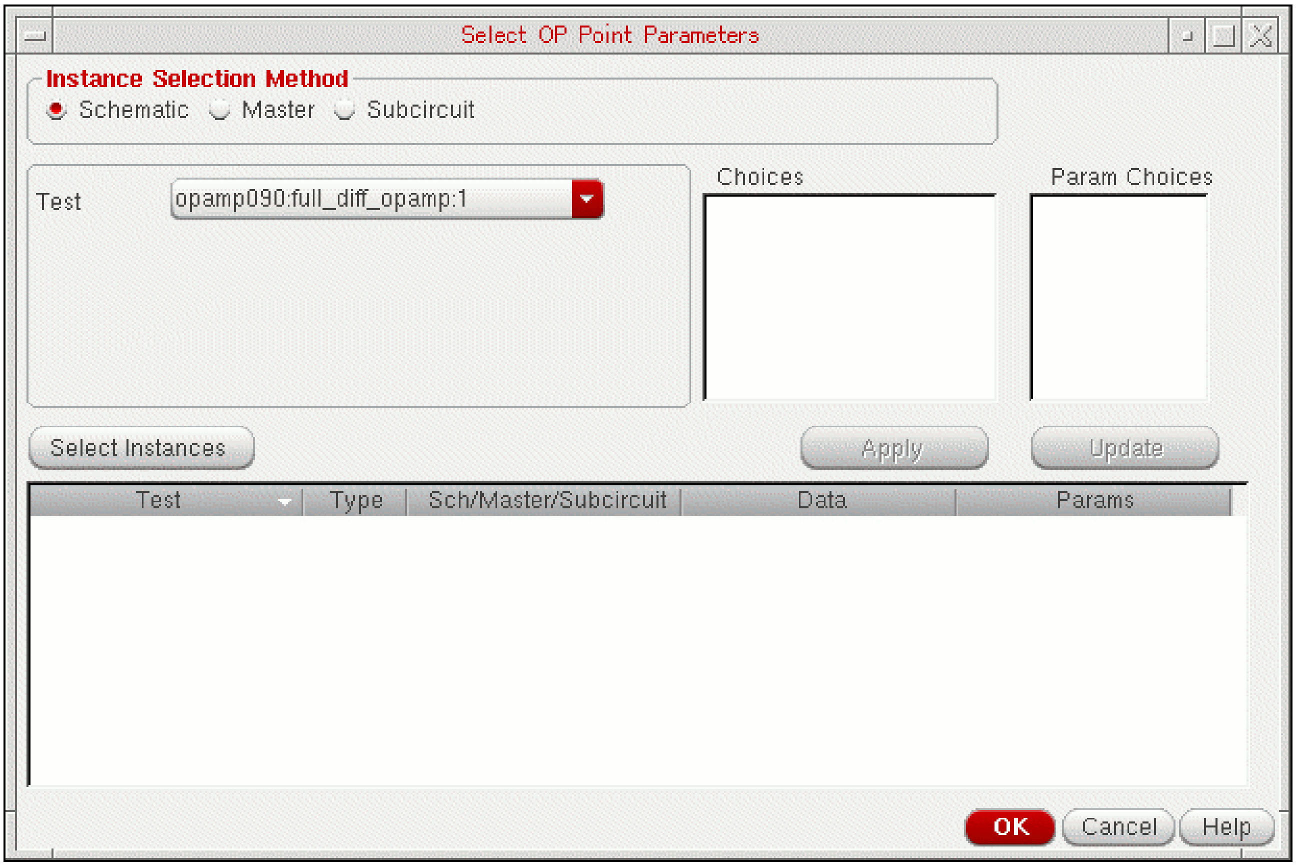
Saving Sensitivity Data for DC Operating Point Parameters

By default, you cannot view the sensitivity data for DC operating point parameters because the data is not saved in the results database. The Select OP Point Parameters form allows you to specify the DC operating point parameters for which you want to save and view sensitivity data. For more information about viewing sensitivity data, see Viewing Sensitivity Data on page 48.

To specify the DC operating point parameters for which you view sensitivity data, do the following:

1. On the Sensitivity form click the *Specify DC Operating Point Parameters* button.

   The Select OP Point Parameters form appears.



2. Do one of the following:

Select	**To**
Schematic	Add DC operating point parameters by selecting instances on the schematic. For more information, see Adding DC Operating Point Parameters of Schematic Instances on page 36.
Master	Add DC operating point parameters of instances of cellviews. For more information, see Adding DC Operating Point Parameters of Cellview Instances on page 40.

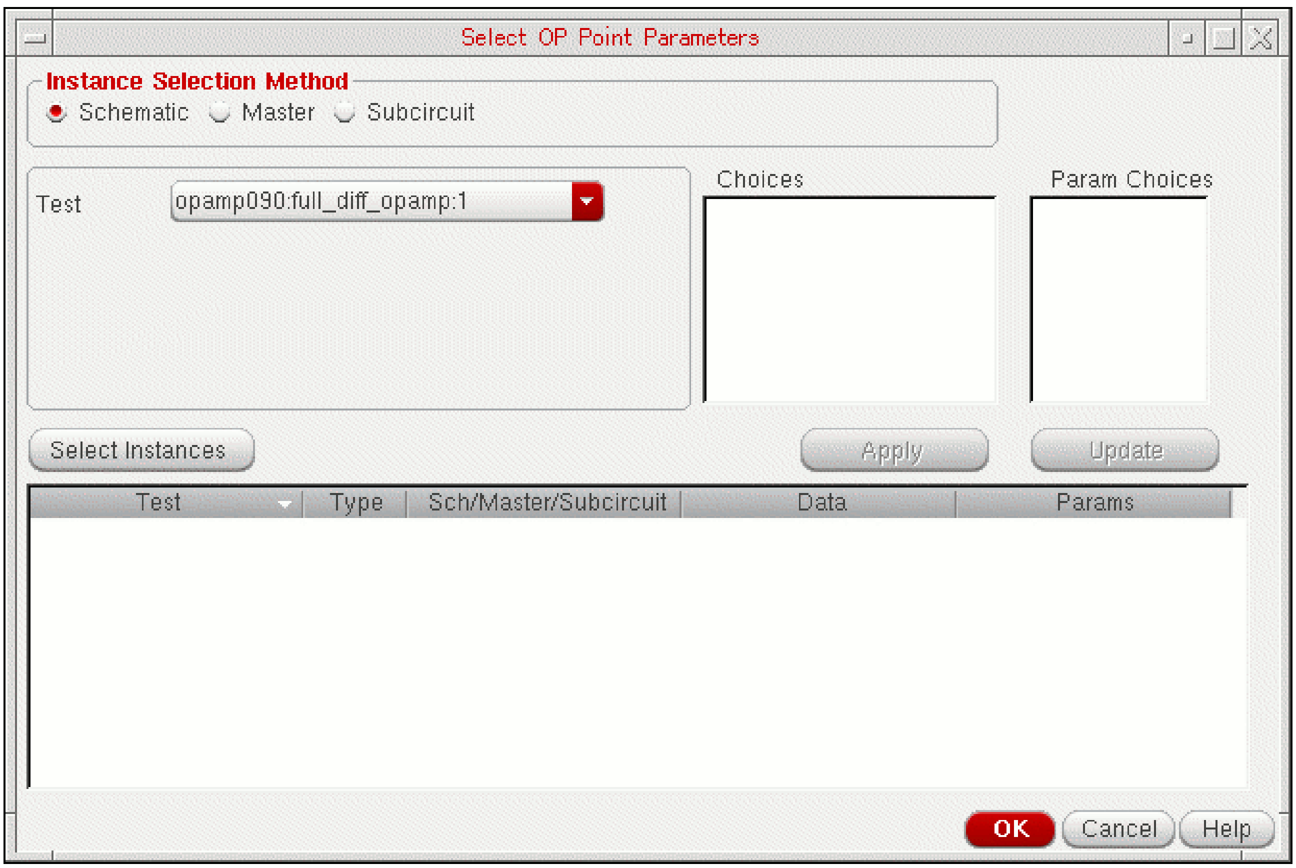
Select	**To**
Subcircuit	Add DC operating point parameters of subcircuit instances. For more information, see Adding DC Operating Point Parameters of Subcircuit Instances on page 44.

3. Click *OK* to save the changes and return to the Sensitivity options form.

Adding DC Operating Point Parameters of Schematic Instances

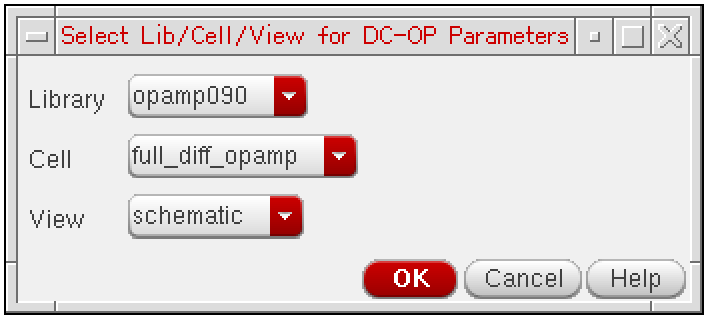
To add DC operating point parameters by selecting instances on the schematic, do the following:

1. Select the *Schematic* option.



2. In the *Test* drop-down list, choose the test in which you want to select instances.

To select instances for all the tests that are enabled in the Data View pane, choose *All Tests* in the *Test* drop-down list. If you choose *All Tests*, the Select Lib/Cell/View for DC-OP Parameters form appears.



Do the following to open the schematic view from which you want to select instances for all the tests that are enabled in the Data View pane.

a. Use the *Library*, *Cell* and *View* drop-down lists to select the schematic view.

Note: The *Library*, *Cell* and *View* drop-down lists display only the cellviews that are used in the designs for all the tests.

b. Click *OK*.

The selected schematic view is opened in a new tab.

3. If a test name is selected in the *Test* drop-down list, click the *Select Instances* button. The schematic for the test is opened in a new tab.

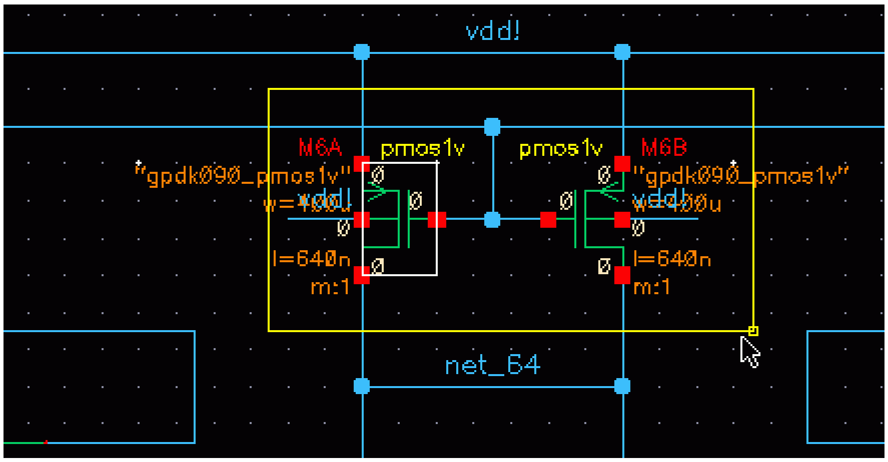
4. In the schematic, select one or more instances.

To select more than one instance at a time, do one of the following:

- ❑ Hold down the *Shift* key and click on instances.
- ❑ Click and drag the mouse over the instances you want to select.

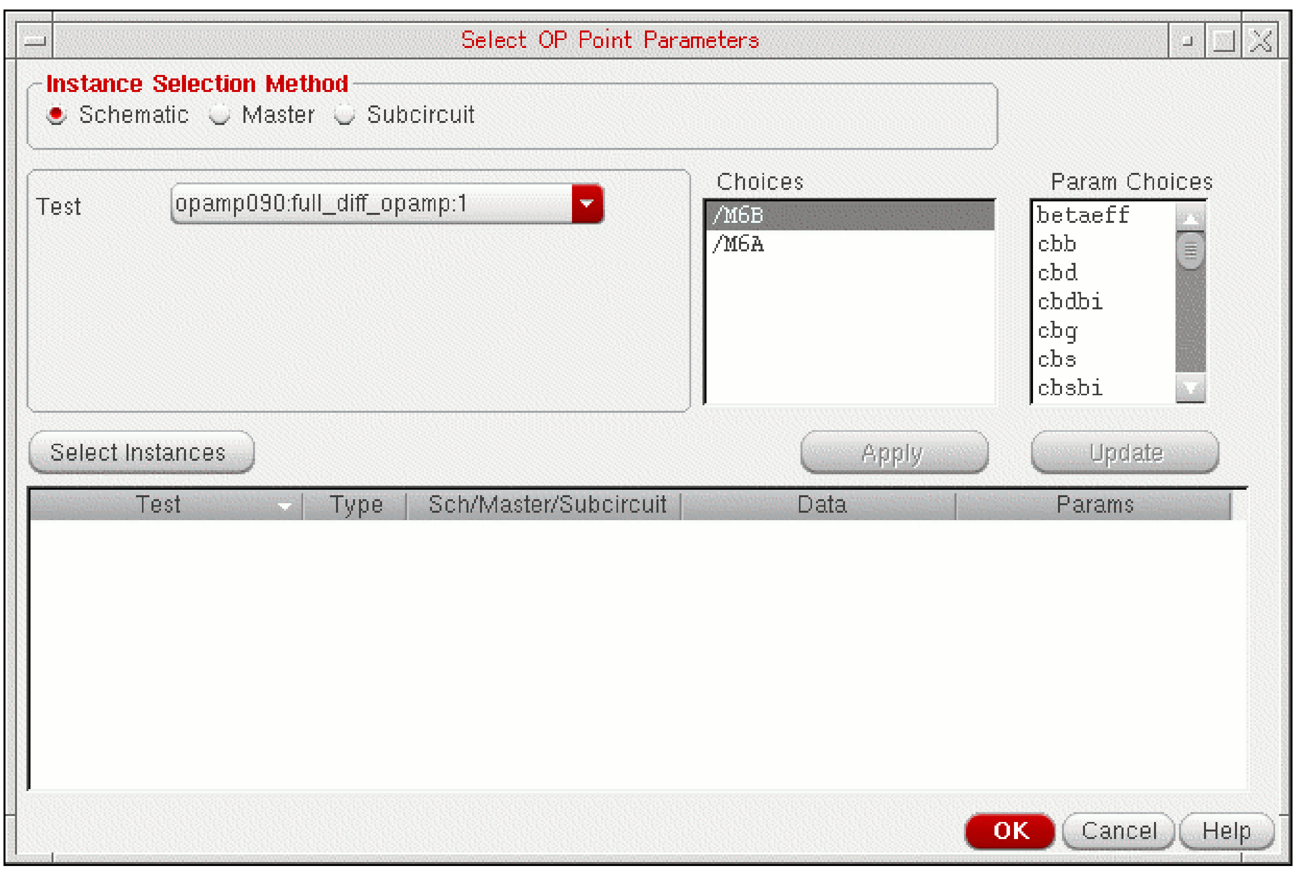
  All the instances that are within the yellow bounding box that appears are included in the selection.

In the following example, instances `M6A` and `M6B` that are within the yellow bounding box are included in the selection.

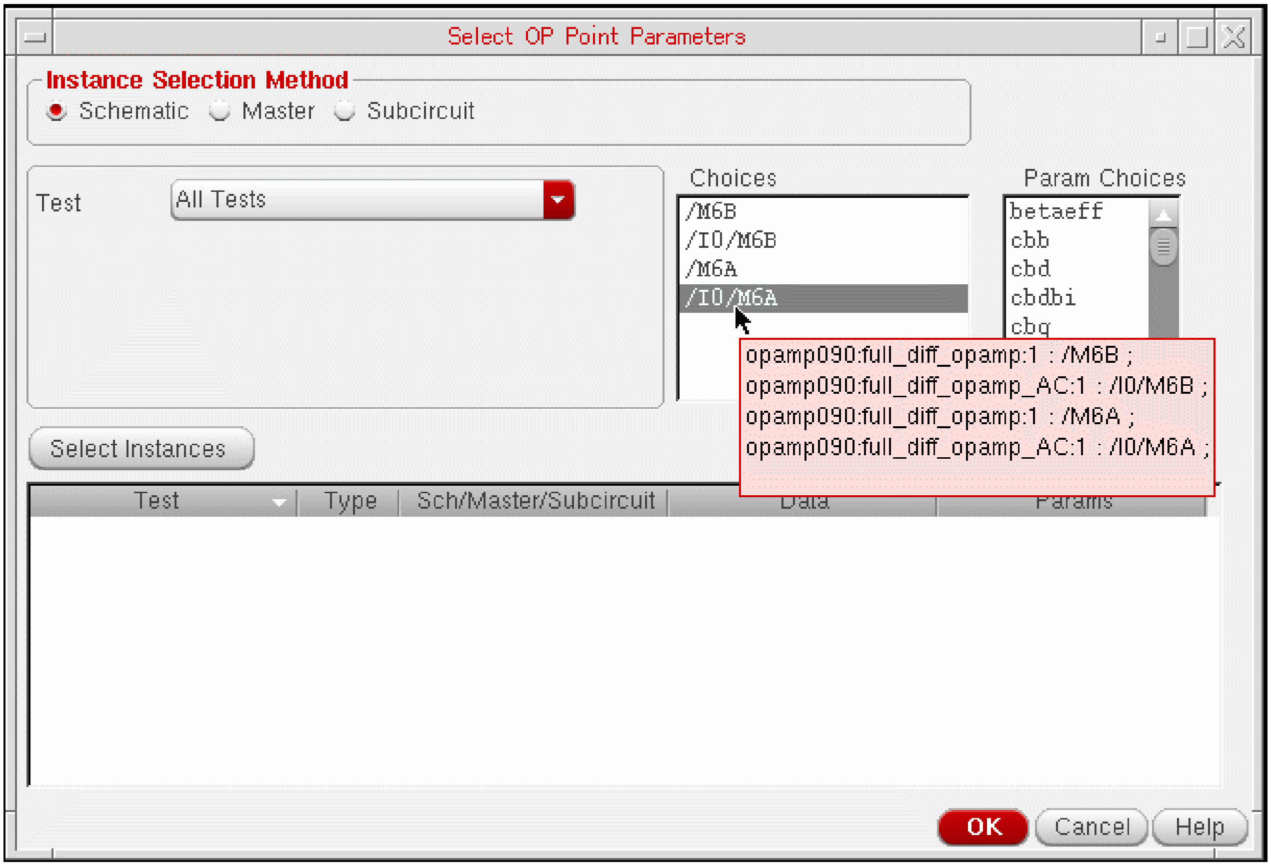


5. Press the *Esc* key when you are done.

The selected instances are displayed in the *Choices* field.



Note: If *All Tests* is selected in the *Test* drop-down list, the instance names of the selected instances in the designs for all the tests are displayed in the *Choices* field. You can place the mouse pointer in the *Choices* field to view the instance names of the selected instances in the design for each test.



6. In the *Choices* field, select the instance for which you want to add DC operating point parameters.

   The DC operating point parameters for the instance are displayed in the *Param Choices* field.

Tip

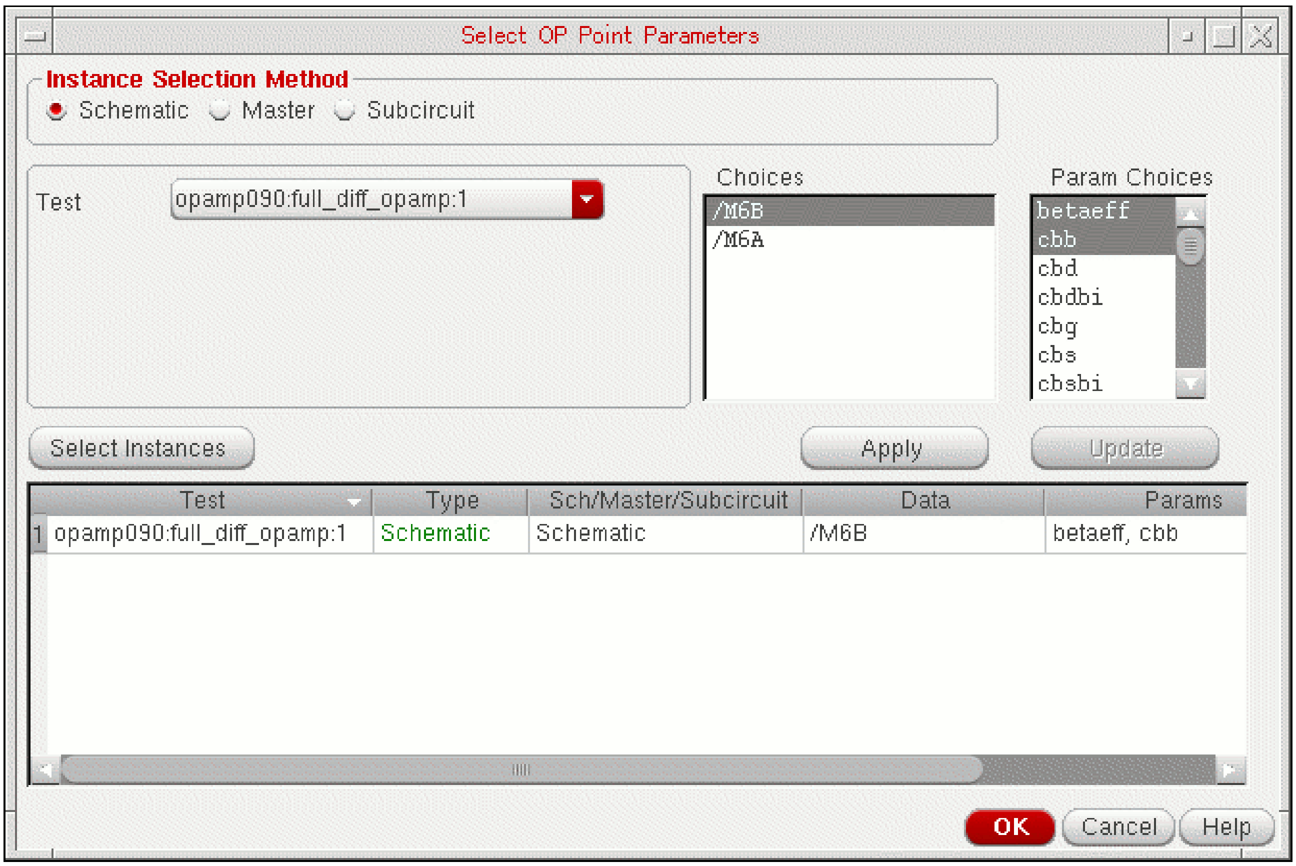
If the same DC operating point parameter exists in more than one instance, you can simultaneously add the parameter for all the instances by selecting those instances in the *Choices* field. To select multiple instances, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next instance to add it to the selection set.

7. In the *Param Choices* field, select the DC operating point parameter you want to add.

To select multiple DC operating point parameters, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next parameter to add it to the selection set.

8. Click *Apply*.

The selected parameters are added for the test. For example, in the following figure, the DC operating point parameters `betaeff` and `cbb` of instance `M6B` are added for the `opamp090:full_diff_opamp:1` test.



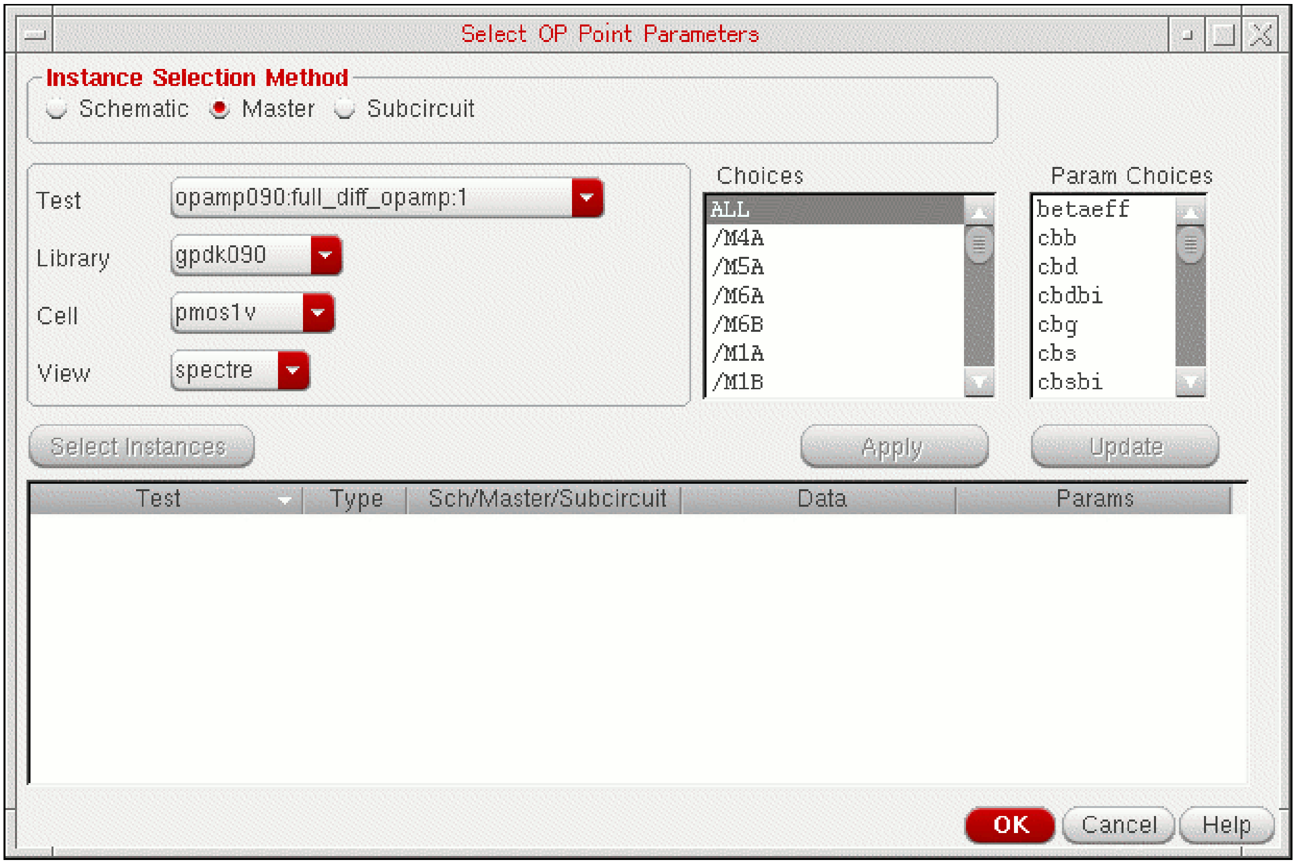
See also:

- Replacing DC Operating Point Parameters on page 47
- Deleting DC Operating Point Parameters on page 47

Adding DC Operating Point Parameters of Cellview Instances

To add DC operating point parameters of instances of cellviews, do the following:

1. Select the *Master* option.



2. In the *Test* drop-down list, choose the test for which you want to select instances of cellviews.

   To select instances for all the tests that are enabled in the Data View pane, choose *All Tests* in the *Test* drop-down list.

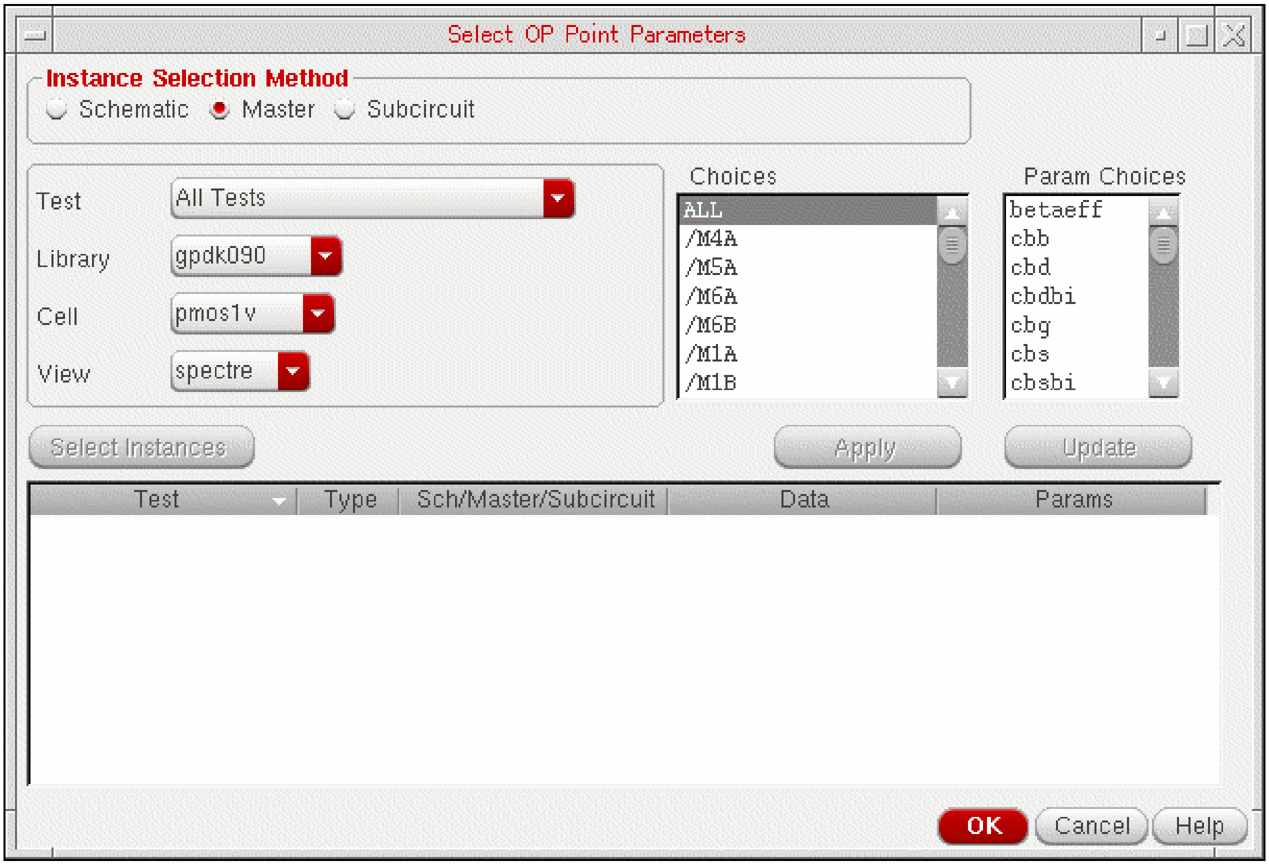
3. Use the *Library*, *Cell* and *View* drop-down lists to select the library, cell and view in which the cellview exists.

   All the instances of the cellview in the design for the test are displayed in the *Choices* field.

   Note the following:

   - The *Library*, *Cell* and *View* drop-down lists display only the libraries and cells for the cellviews that have DC operating point parameters.

   - If *All Tests* is selected in the *Tests* drop-down list, the *Choices* field displays the instance names of the cellview in the designs for all the tests that are enabled in the Data View pane. For example, in the following figure, the *Choices* field displays the instance names of the `spectre` cellview of the `pmos1v` cell in the designs for all the tests.



Note: You can place the mouse pointer in the *Choices* field to view the instance name of the cellview in the design for each test.

4. In the *Choices* field, select the instance for which you want to add DC operating point parameters.

The DC operating point parameters for the instance are displayed in the *Param Choices* field.

Note the following:

- ❑ To add DC operating point parameters for all instances of the cellview, select *ALL* in the *Choices* field.
- ❑ If the same DC operating point parameter exists in more than one instance, you can simultaneously add the parameter for all the instances by selecting those instances in the *Choices* field. To select multiple instances, hold down the *Shift* key (for

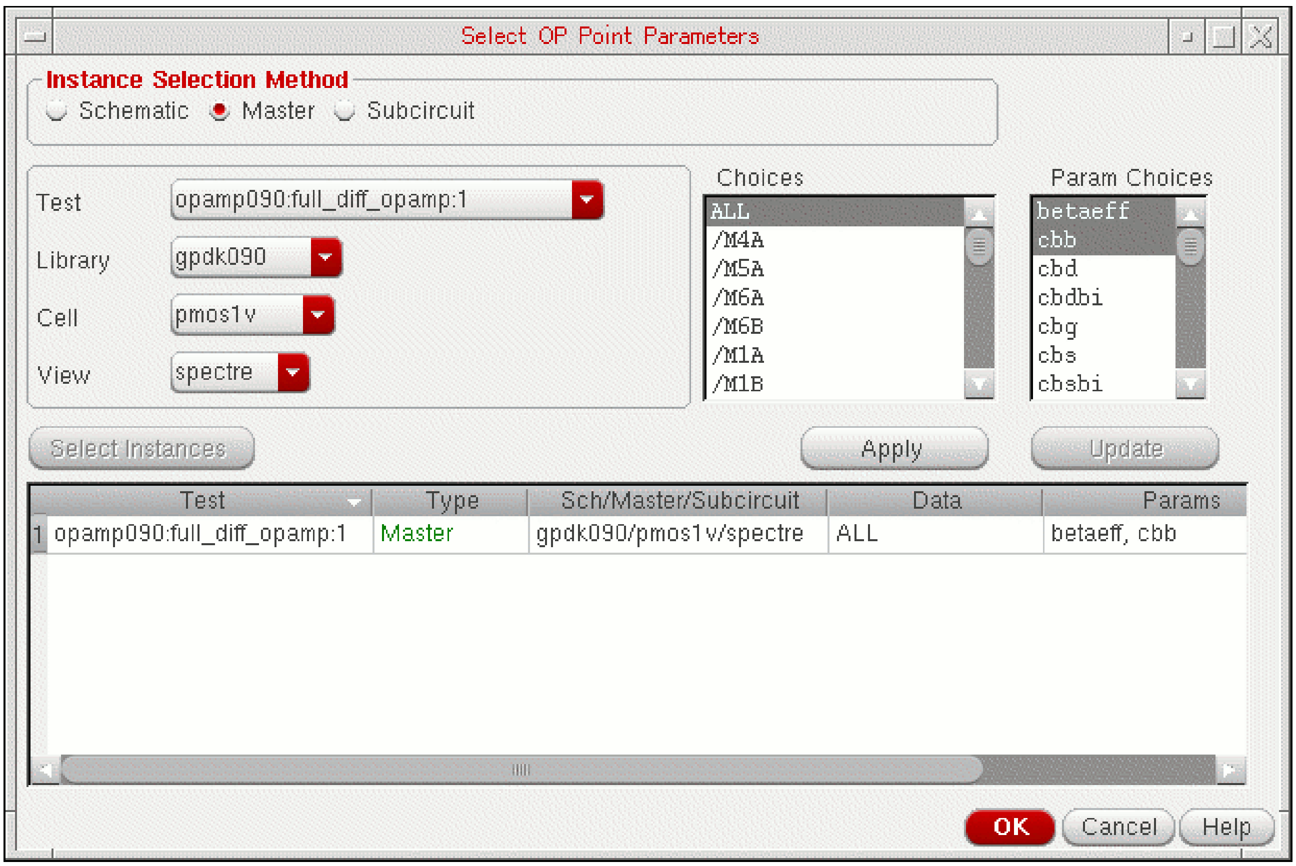
contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next instance to add it to the selection set.

5. In the *Param Choices* field, select the DC operating point parameter you want to add.

   To select multiple DC operating point parameters, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next parameter to add it to the selection set.

6. Click *Apply*.

   The selected parameters are added for the test. For example, in the following figure, the DC operating point parameters `betaeff` and `cbb` on all instances of the `spectre` cellview of the `pmos1v` cell are added for the `opamp090:full_diff_opamp:1` test.



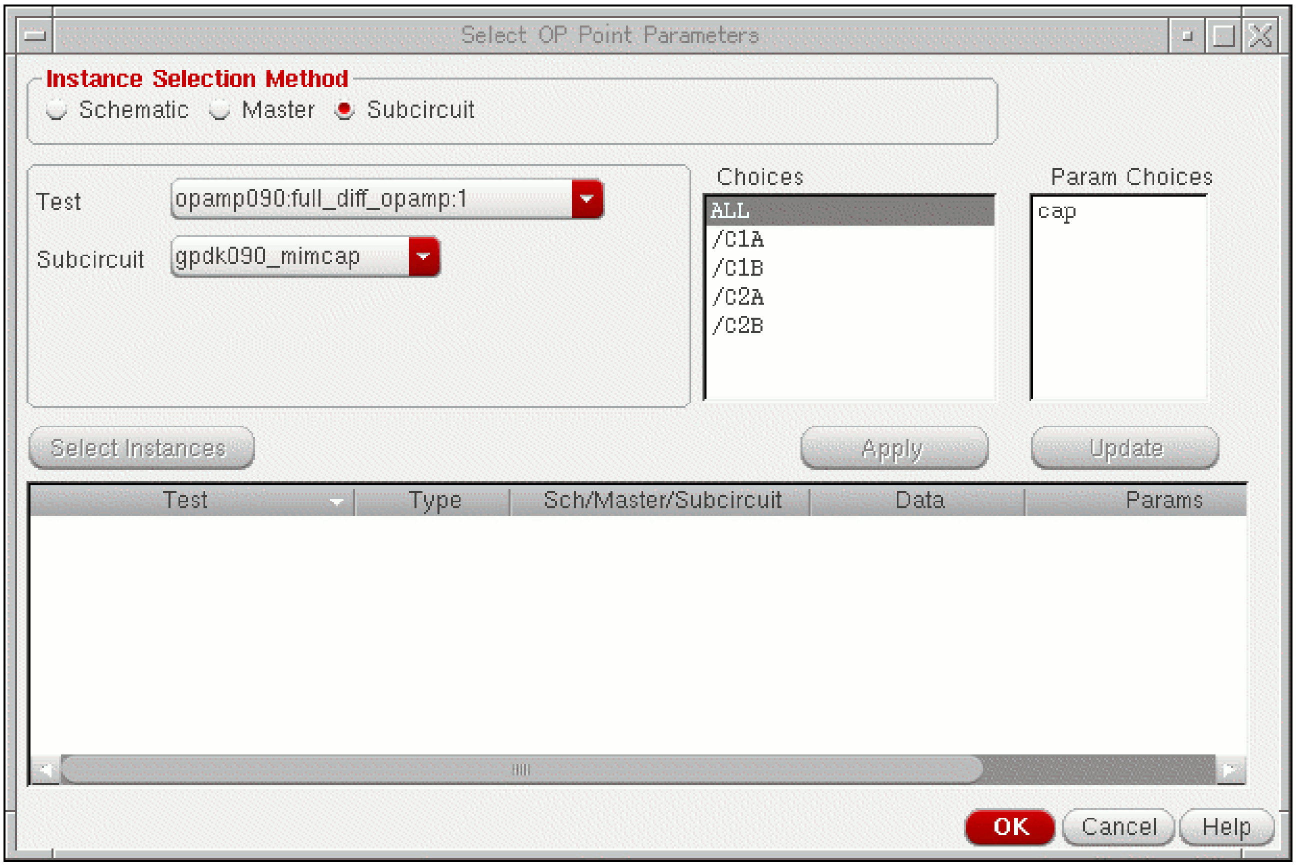
See also:

- Replacing DC Operating Point Parameters on page 47
- Deleting DC Operating Point Parameters on page 47

Adding DC Operating Point Parameters of Subcircuit Instances

To add DC operating point parameters of subcircuit instances, do the following:

1. Select the *Subcircuit* option.



2. In the *Test* drop-down list, choose the test for which you want to select subcircuit instances.

   To select subcircuit instances for all the tests that are enabled in the Data View pane, choose *All Tests* in the *Test* drop-down list.

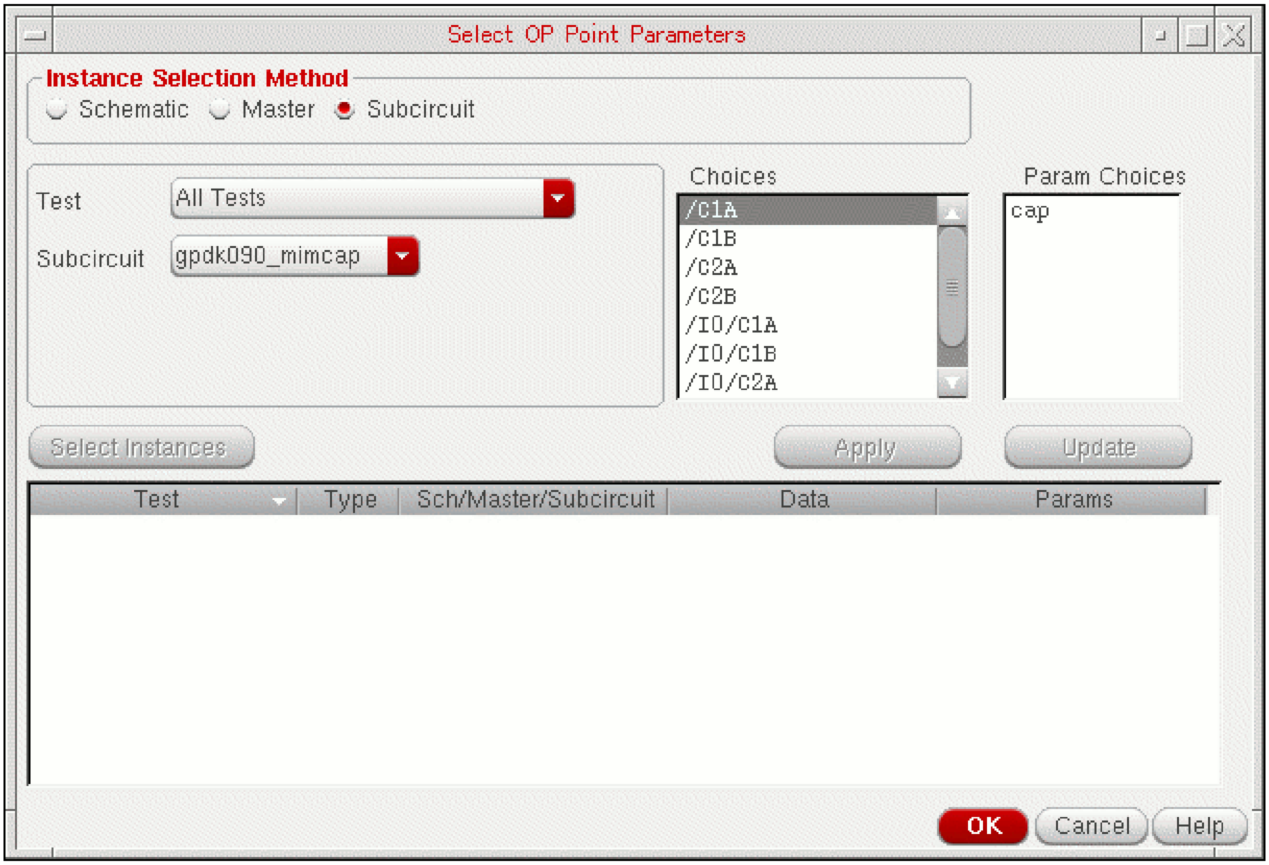
3. In the *Subcircuit* drop-down list, choose the subcircuit whose instances you want to select.

   All the instances of the subcircuit in the design are displayed in the *Choices* field.

   Note the following:

   - The *Subcircuit* drop-down list displays only the subcircuits that have DC operating point parameters.

- If *All Tests* is selected in the *Tests* drop-down list, the *Choices* field displays the instance names of the subcircuit in the designs for all the tests. For example, in the following figure, the *Choices* field displays the instance names of the `ampn` subcircuit in the designs for all the tests.



You can place the mouse pointer in the *Choices* field to view the instance name of the subcircuit in the design for each test.

4. In the *Choices* field, select the instance for which you want to add DC operating point parameters.

   The DC operating point parameters for the instance are displayed in the *Param Choices* field.

Tip

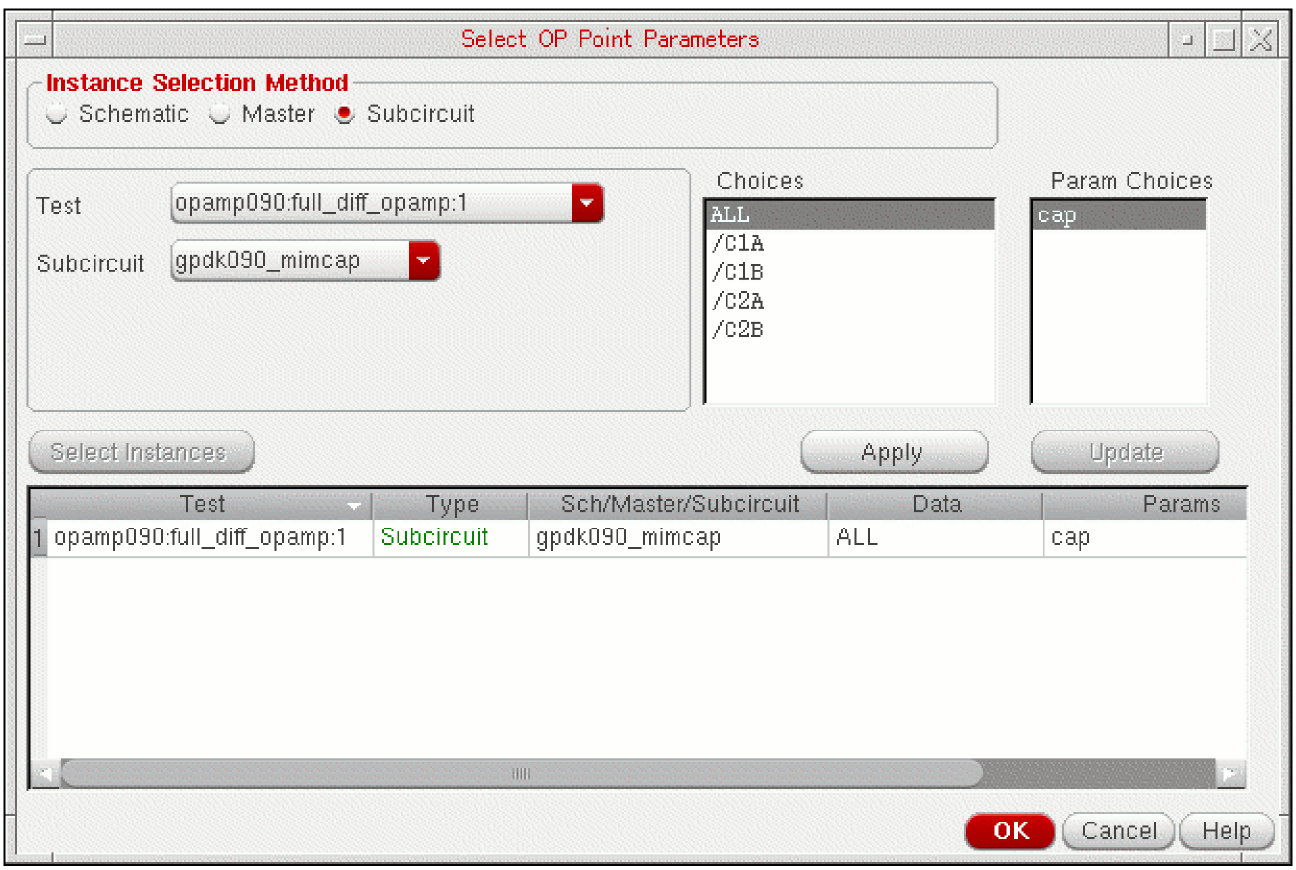
If the same DC operating point parameter exists in more than one instance, you can simultaneously add the parameter for all the instances by selecting those instances in the *Choices* field. To select multiple instances, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next instance to add it to the selection set.

5. In the *Param Choices* field, select the DC operating point parameter you want to add.

To select multiple DC operating point parameters, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection) and click the next parameter to add it to the selection set.

6. Click *Apply*.

The selected parameters are added for the test. For example, in the following figure, the DC operating point parameter `cap` on all instances of the `gpdk090_mimcap` subcircuit are added for the `opamp090:full_diff_opamp:1` test.



See also:

- Replacing DC Operating Point Parameters on page 47
- Deleting DC Operating Point Parameters on page 47

Replacing DC Operating Point Parameters

To replace one DC operating point parameter with another, do the following:

1. Select the row for the parameter.
2. Select a parameter in the *Param Choices* field.
3. Click the *Update* button.

   The parameter in the row is replaced with the parameter selected in the *Param Choices* field.

Deleting DC Operating Point Parameters

To delete DC operating point parameters, do the following:

➡ Right-click on the row for a parameter and choose *Delete*.

To delete multiple parameters, hold down the *Shift* key (for contiguous selection) or the *Ctrl* key (for noncontinguous selection), click the rows for the parameters you want to delete, then right-click and choose *Delete*.

To delete all parameters, do the following:

1. Right-click on the row for a parameter and choose *Select All*.
2. Right-click on the row for a parameter and choose *Delete*.

Viewing Sensitivity Data

After the Sensitivity Analysis or the Monte Carlo run is complete, ADE GXL automatically opens the Sensitivity Analysis window with sensitivity analysis results displayed in it.

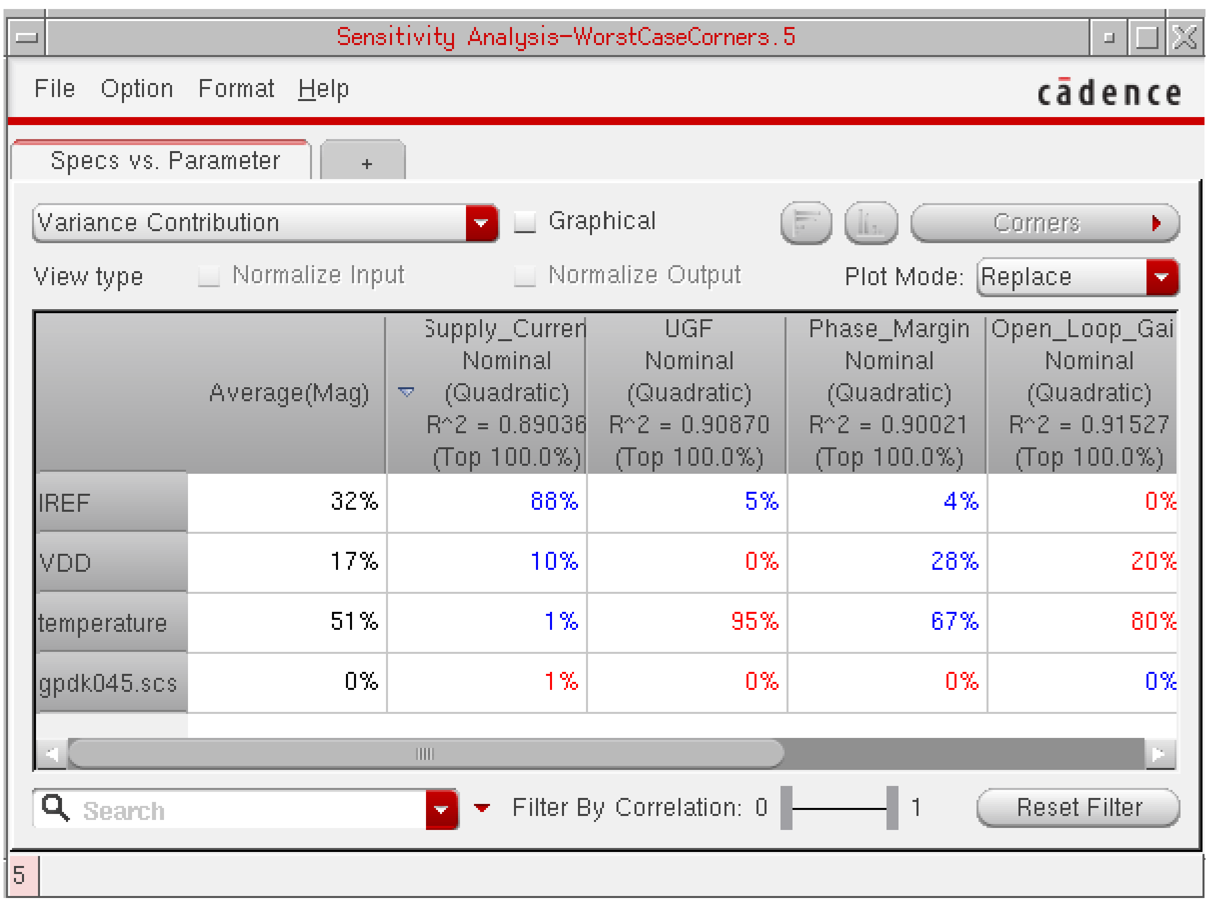
To view sensitivity data from a previous sensitivity analysis run, do one of the following:

➡ Click the *Sensitivity Results* button on the Results tab.

➡ In the *History* tab on the Data View pane, right-click on the history item for a Sensitivity Analysis or Monte Carlo run and choose *Sensitivity Results*.

The Sensitivity Analysis window appears.

Figure 2-1 Sensitivity Analysis Window



Reading Data in the Sensitivity Analysis window

In the Sensitivity Analysis window, sensitivity data is displayed in a tabular format, where each data cell shows the sensitivity of a spec or parameter in the corresponding column to the spec or parameter displayed in the corresponding row. The specs or parameters displayed in the inputs and outputs depend on the selected tab.

By default, the data shows sensitivity of each specification with respect to the available device and statistical parameters. This view is governed by the *Specs vs. Parameter* tab. To know more about the other available tabs, refer to Viewing Sensitivity Data in Different Tabs.

If the analysis was run on various corners, a column is added for each corner condition. For example, in the figure shown above, data for the `gainBwProd` spec is shown for all the three corners, `nominal`, `C0_0`, and `C0_2`.

Note: If you disable any test for a corner in the **Corners Setup** form, specification columns for that test and corner combination are not shown in the Sensitivity Analysis window.

There are various ways in which you can customize the way in which the sensitivity data is displayed. For more details on how to customize the view, refer to Working with Sensitivity Data.

For details on how to plot sensitivity results, refer to Plotting Sensitivity Analysis Results.

Viewing Sensitivity Data in Different Tabs

The Sensitivity Analysis window provides the following tabs for viewing sensitivity data:

- Specs vs. Specs
- Specs vs. Parameter
- Specs vs. Stat Param
- Specs vs. DC Ops
- DC Ops vs. Parameter
- DC Ops vs. Stat Param
- Stat Param vs. Stat Param

In these tabs, the input factor is displayed in the rows and the output is displayed in the columns. For example, in the Specs vs. Parameter tab shown in Figure 2-1 on page 48, parameters are input factors and specifications are the output. Therefore, parameters are displayed in the rows and specifications are displayed in the columns. For information about working with the data in the tabs, see Working with Sensitivity Data on page 53.

Note: The text `N.A.` in a cell in the Sensitivity Analysis window indicates that the sensitivity value could not be calculated because of some reason. You can place the mouse pointer on the cell to view the reason in a tooltip.

These tabs are described below:

Tab Name	Description
Specs vs. Specs	Displays the sensitivity of each specification with respect to other specifications. Do the following to display the tab: **1.** From the drop-down list, choose *Specification vs. Specification*. **2.** Click the + button to display the Specs vs. Specs tab.
Specs vs. Parameter	Displays the sensitivity of each specification with respect to device and statistical parameters. Do the following to display the tab: **1.** From the drop-down list, choose *Specification vs. Parameter*. **2.** Click the + button to display the Specs vs. Parameter tab. **Note:** The sensitivity of statistical parameters is displayed in this tab only if sensitivity data for statistical parameters exists in the results database. See also, Hiding or Showing Statistical Parameters on page 73.

<table>
<tr><th>Tab Name</th><th>Description</th></tr>
<tr><td>Specs vs. Stat Param</td><td>Displays the sensitivity of each specification with respect to statistical parameters.

Do the following to display the tab:

1. From the drop-down list, choose Specification vs. Statistical Parameter.

Note: The Specification vs. Statistical Parameter option is available only if sensitivity data for statistical parameters exists in the results database.

2. Click the + button to display the Specs vs. Stat Param tab.</td></tr>
<tr><td>Specs vs. DC Ops</td><td>Displays the sensitivity of each specification with respect to DC operating point parameters.

Do the following to display the tab:

1. From the drop-down list, choose Specification vs. DC OP Point Parameter.

Note: The Specification vs. DC OP Point Parameter option is available only if sensitivity data for DC operating point parameters exists in the results database.

2. Click the + button to display the Specs vs. DC Ops tab.</td></tr>
<tr><td>DC Ops vs. Parameter</td><td>Displays the sensitivity of each DC operating point parameter with respect to device and statistical parameters.

Do the following to display the tab:

1. From the drop-down list, choose DC OP Point Parameter vs. Parameter.

Note: The DC OP Point Parameter vs. Parameter option is available only if sensitivity data for DC operating point parameters exists in the results database.

2. Click the + button to display the DC Ops vs. Parameter tab.

Note: The sensitivity of statistical parameters is displayed in this tab only if sensitivity data for statistical parameters exists in the results database. See also, <u>Hiding or Showing Statistical Parameters</u> on page 73.</td></tr>
</table>

<table>
<tr><th>Tab Name</th><th>Description</th></tr>
<tr><td>DC Ops vs. Stat Param</td><td>Displays the sensitivity of each DC operating point parameters with respect to statistical parameters.

Do the following to display the tab:

1. From the drop-down list, choose DC OP Point Parameter vs. Statistical Parameter.

Note: The DC OP Point Parameter vs. Statistical Parameter option is available only if sensitivity data for DC operating point parameters and statistical parameters exists in the results database.

2. Click the + button to display the DC Ops vs. Stat Param tab.</td></tr>
<tr><td>Stat Param vs. Stat Param</td><td>Displays the sensitivity of each statistical parameter with respect to other statistical parameters.

Do the following to display the tab:

1. From the drop-down list, choose Statistical Parameter vs. Statistical Parameter.

Note: The Statistical Parameter vs. Statistical Parameter option is available only if sensitivity data for statistical parameters exists in the results database.

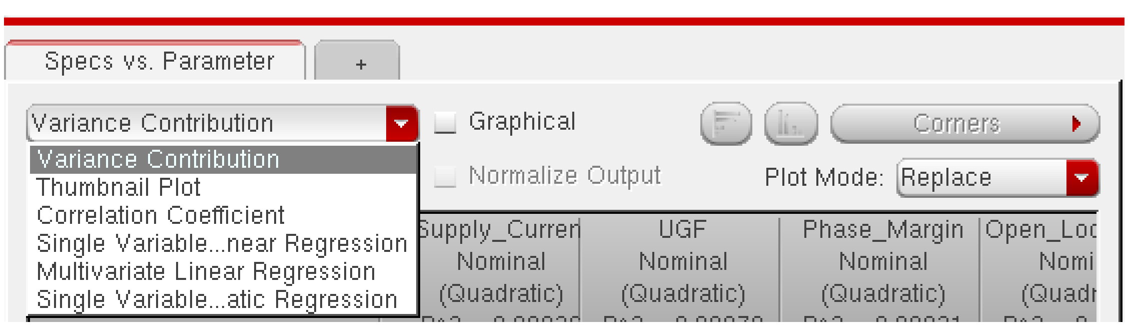
2. Click the + button to display the Stat Param vs. Stat Param tab.</td></tr>
</table>

Working with Sensitivity Data

This following topics describe how you can work with the data in the Sensitivity Analysis window:

Changing the Type and Format of Data To View

By default, the Sensitivity Analysis window shows the Variance Contribution data in percentage format. You can also view an overview of the data in thumbnail form or view the correlation and regression data by selecting an option from the drop-down list.



By default, the Sensitivity Analysis window shows both correlation and regression data in the percentage format. This Sensitivity Analysis window is displayed when you analyze data by using the one-factor-at-a-time (OFAT) method.

When you analyze data during a Monte Carlo or Hammersley sampling run, you will notice an add it on al format, *Multivariate Linear Regression*, in the drop-down list displayed in the figure above.

In the one-factor-at-a-time (OFAT) method, a single variable linear regression model is followed, which means that an individual independent variable is created for each row in the sensitivity table. However, in the multivariate Linear Regression method, one common model is applied to all the rows in the sensitivity table.

To change the format of data to the graphical format:

➤ Select the *Graphical* checkbox next to the drop-down list at the top of a tab.

For more details on different types and formats of data in the Sensitivity Analysis window, see:

- Viewing Correlation Data on page 56
- Viewing Correlation Data in Graphical Format on page 57
- Viewing Thumbnail Plots on page 59
- Viewing Regression Data on page 61
- Viewing Normalized Regression Data on page 63
- Viewing Regression Data in Graphical Format on page 66

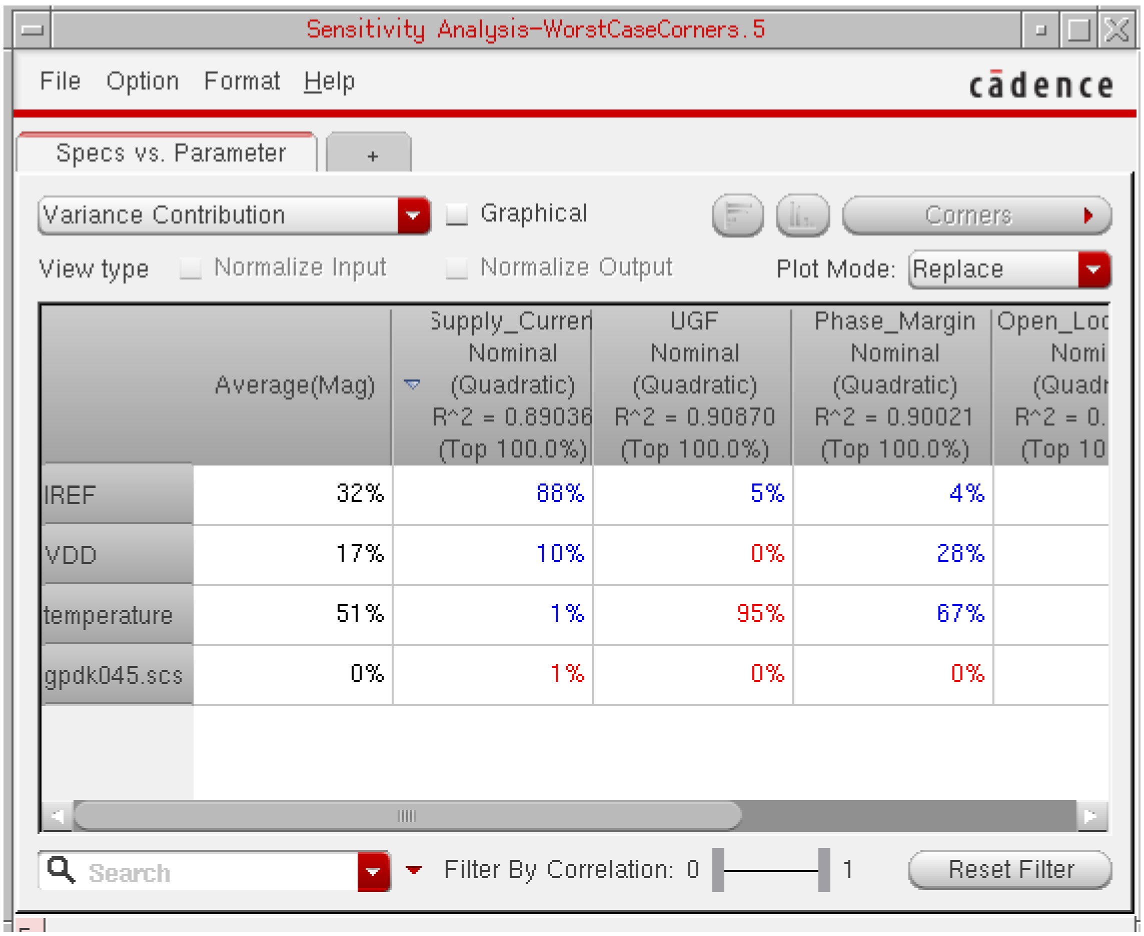
Viewing Variance Contribution Data

Variance Contribution uses a similar method as used by Mismatch Contribution (refer to Analyzing the Mismatch Contribution on page 194). Note that Mismatch Contribution is applied to Monte Carlo results, where as Variance Contribution is used primarily to view the Sensitivity Analysis results; however, it can also be applied to a Single run, Sweeps and Corners runs, and Optimization runs.

To view the variance contribution data:

➤ Choose `Variance Contribution` from the drop-down list at the top of a tab.

The variance contribution data appears as shown in the figure below. By default, this data is displayed in the percentage format.

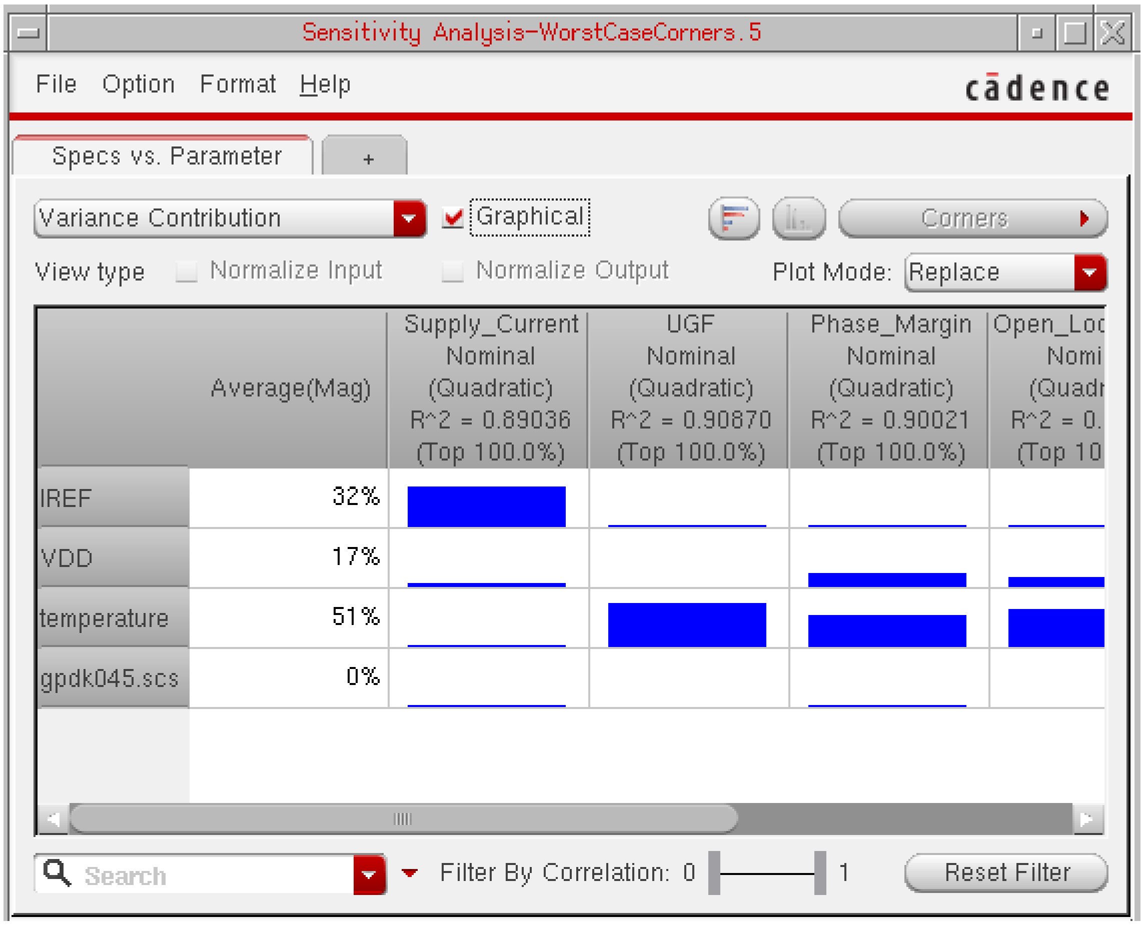


In the above figure, the first column lists the variables, and the *Average(Mag)* column displays the average of percentage values of the variance contribution for each specification. Each cell in the other columns in the table displays the variance contribution values. Positive values are displayed in blue color and negative values are displayed in red color.

Viewing Variance Data in Graphical Format

When the *Graphical* check box is selected, the variance contribution data is displayed as shown in the following figure.

Figure 2-2 Variance Data Displayed in the Graphical Format



The first column lists the variables, and the *Average(Mag)* column displays the average of the fit values for each sensitivity in the row. Each cell in the other columns in the table displays a symbol that portrays:

- whether the relationship is positive or negative
- how tightly the data fits to a line

Viewing Correlation Data

Correlation data tells how correlated each goal is to that variable. If the correlation is close to 1 (or -1), then they are highly correlated. However, if the correlation is 0, then the goal and variable are not correlated. The correlation value is always between -1 and 1.

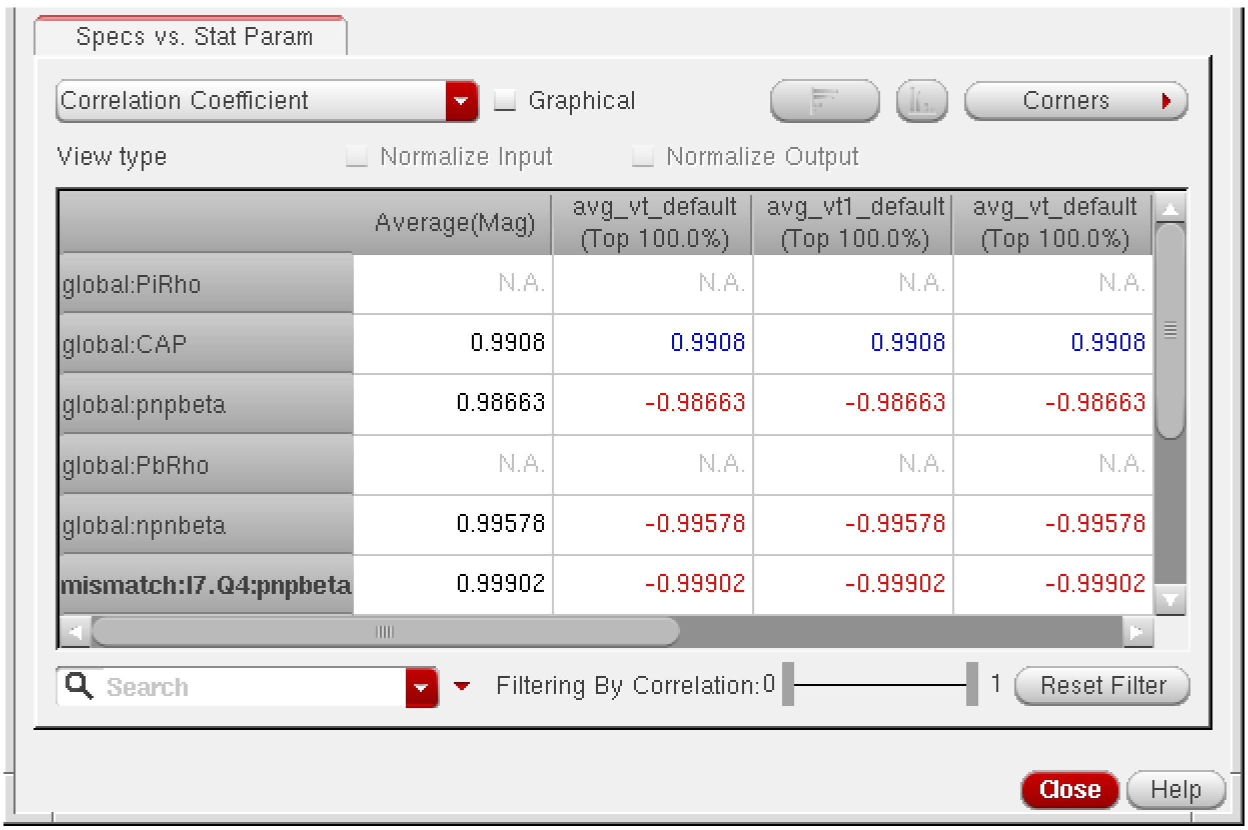
To view correlation data:

➤ Choose `Correlation Coefficient` from the drop-down list at the top of a tab.

The correlation data appears. By default, this data is in the graphical format. For more details on this, refer to Viewing Variance Data in Graphical Format on page 55. To view the correlation data in numeral format, clear the *Graphical* checkbox.

Correlation data in numeral format is displayed as shown in the following figure.

Figure 2-3 Correlation Data in Sensitivity Analysis window

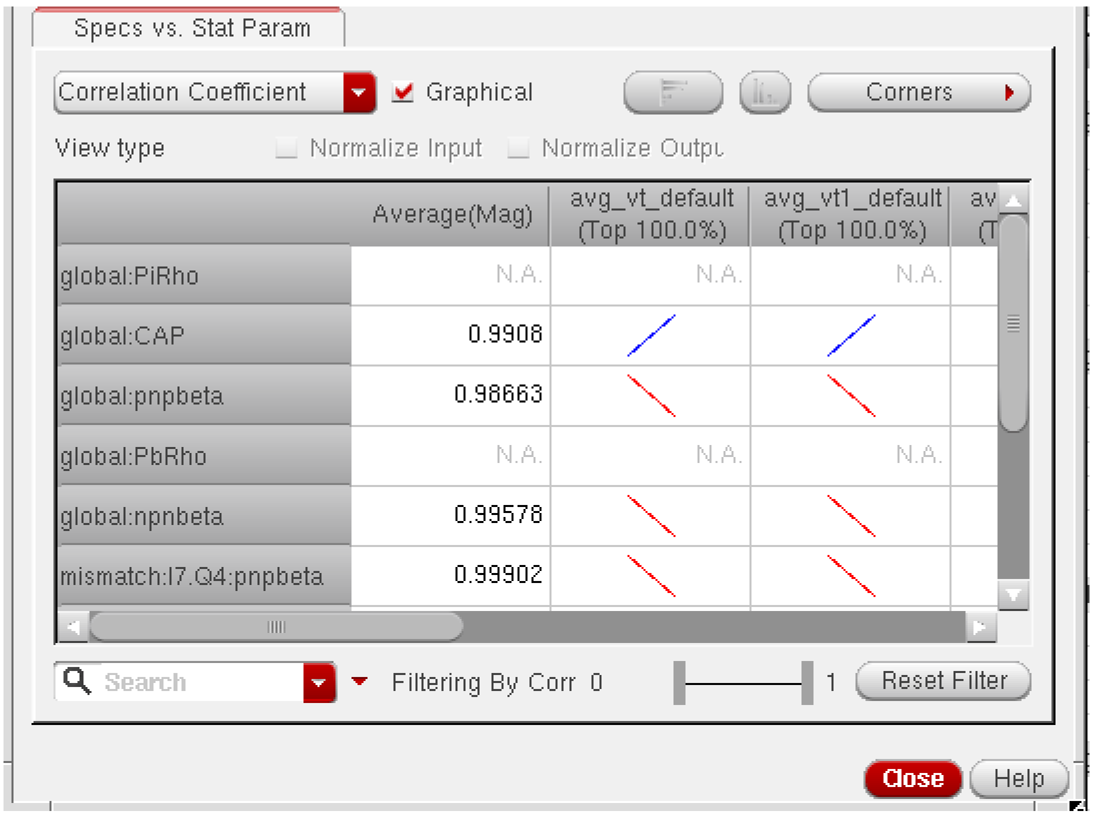


The first column lists the variables, and the *Average(Mag)* column displays the average of absolute values of correlation coefficients for each specification. Each cell in the other columns in the table displays the correlation values. Positive correlation values are displayed in blue color and negative values are displayed in red color.

Viewing Correlation Data in Graphical Format

When the *Graphical* checkbox is selected, the correlation data is displayed as shown in the following figure.

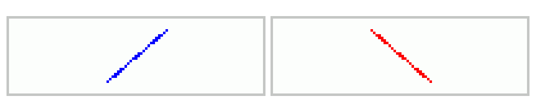
Figure 2-4 Correlation Data Displayed in the Graphical Format



The first column lists the variables, and the *Average(Mag)* column displays the average of the fit values for each sensitivity in the row. Each cell in the other columns in the table displays a symbol that portrays:

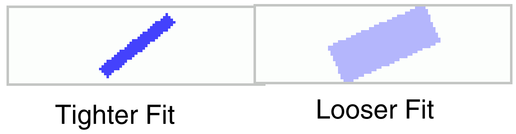
- whether the relationship is positive or negative
- how tightly the data fits to a line

The polarity of the relationship is indicated by the direction of the slope of the line. A blue line that slopes from the lower left to upper right is positive. A red line that slopes from the upper left to the lower right is negative.



Positive Negative

ADE GXL illustrates how tightly the data fits to a line using the width and intensity of the shape. The narrower and brighter the line, the tighter the fit.

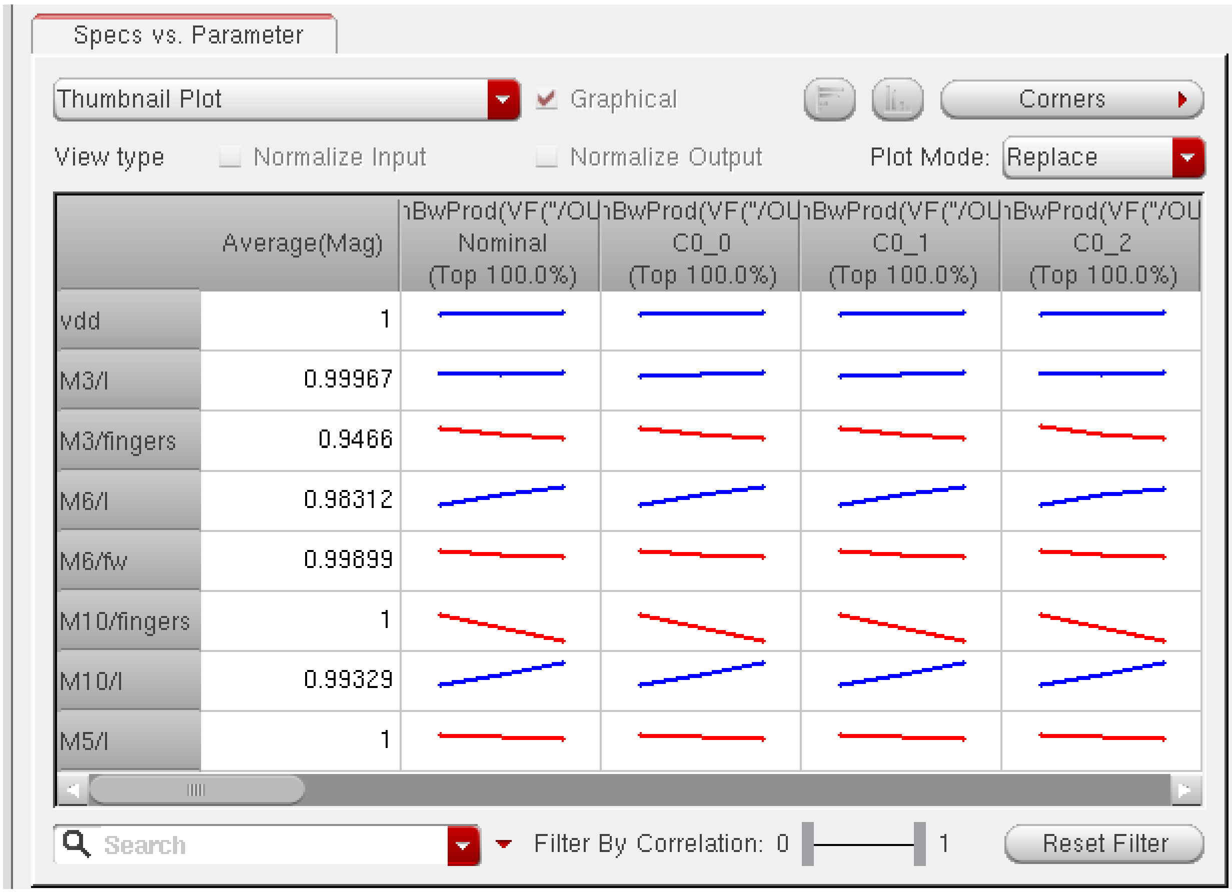


Viewing Thumbnail Plots

The default graphical view is the correlation data, which shows the positive or negative relation between the inputs and outputs. However, it does not show the view of exact relationship between the two. To show a better graphical representation of the relationship between the inputs and outputs, you can choose to show thumbnail images of the actual plots of the Sensitivity Analysis or Monte Carlo run results.

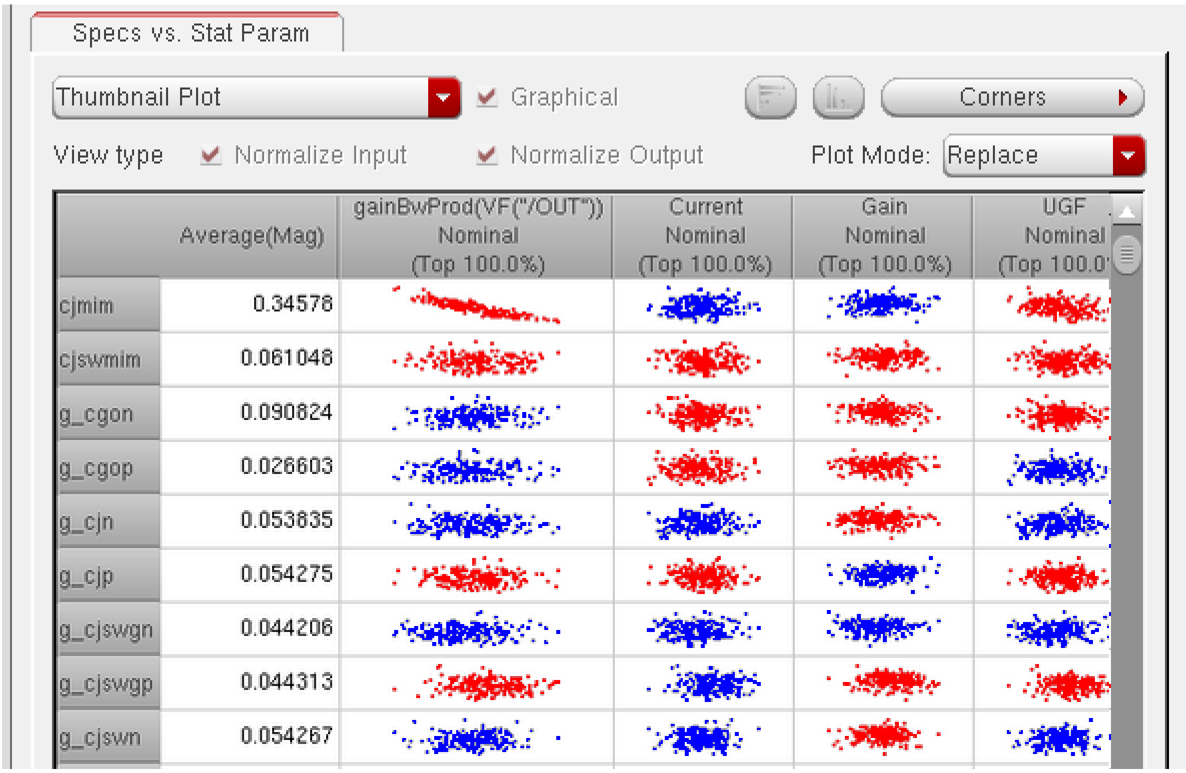
To show thumbnail plots, choose `Thumbnail Plots` from the drop-down list at the top of a tab. With this option, thumbnail plots of the results data are displayed in each cell, as shown in the following figure.

Figure 2-5 Thumbnail Plots with Line Graphs



By default, if the waveform contains ten points or less, a line plot is displayed, as shown in the figure above. For more than ten points, the thumbnail shows scatter plots, as shown in the following figure.

Figure 2-6 Thumbnail Plots with Scatter Graphs



If you want, you can specify the minimum number of points for which you want to display scatter plots by setting the *sensitivityThumbnailMaxPointsForLinePlot* environment variable.

The color of thumbnail plots depends on the sign of correlation coefficients. For positive correlation coefficients, the plots are displayed in blue and for negative values, they are displayed in red.

The thumbnail plot view gives you a view of all the plots at the same time. Click on any thumbnail to view the plot in the Virtuoso Visualization and Analysis XL window.

Viewing Regression Data

The regression data shows the best fit line or the regression line that gives an output value which is very close to the spec value.

The Sensitivity Analysis window allows you to view the following types of regression data.

- The single variable linear regression is the slope of the line between goal values and variable values that best fits the data points. Thus, a change in the variable multiplied by the regression co-efficient is the change that can be expected in the goal value.

  Note: The least square method is used to display the best fit line.

- The single variable quadratic regression uses a quadratic equation to fit the goal values with respect to the variable values. The coefficients from this option can be considered when the correlation values are close to zero and cannot be modelled by a straight line.

- The multivariate regression gives an estimate of the linear relationship between the output and all inputs.

To view regression data, choose one of the following from the drop-down list at the top of a tab:

- `Single Variable Linear Regression` to view single variable linear regression data.

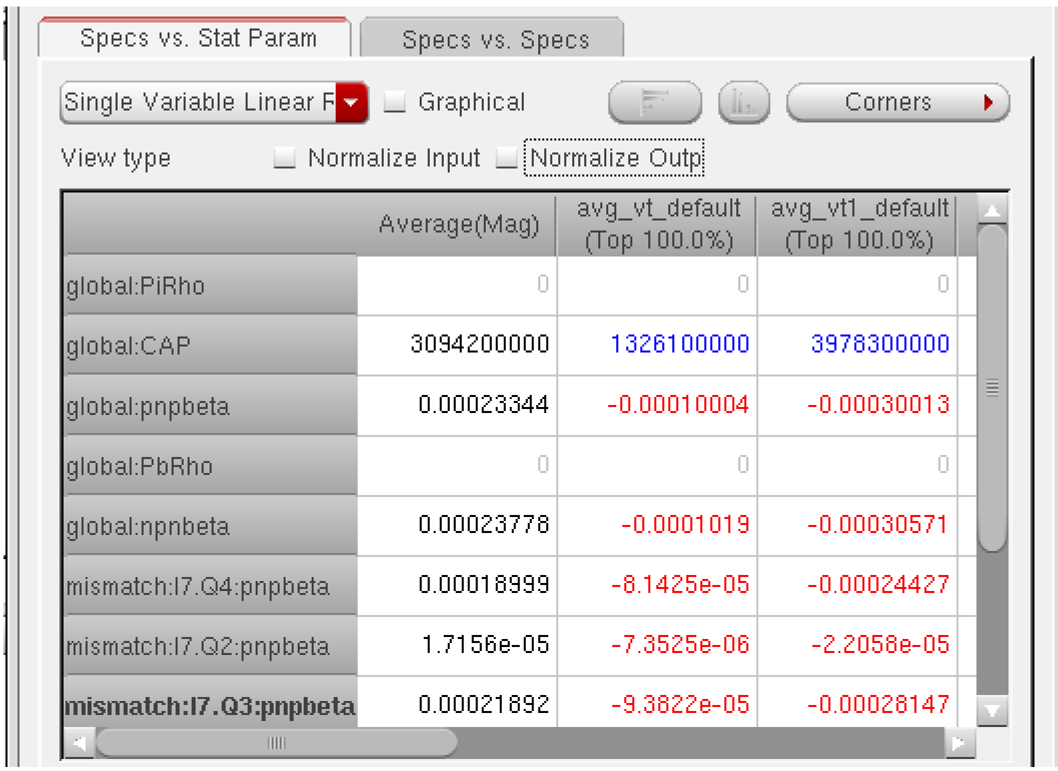
- `Single Variable Quadratic Regression` to view single variable quadratic regression data.

- `Multivariate Linear Regression` to view multivariate linear regression data.

  By default, the regression data appears in the graphical format. For more details, refer to Viewing Regression Data in Graphical Format on page 66.

  Clear the *Graphical* checkbox to view the data in numeral format. The regression data appears as shown in the following figure.

Figure 2-7 Regression Data in Sensitivity Analysis window



The first column lists the variables, and the *Average(Mag)* column displays the average of absolute values of regression coefficients for each specification. Each cell in the other columns in the table displays the regression values. Positive regression values are displayed in blue color and negative values are displayed in red color.

Viewing Normalized Regression Data

Normalized regression is the value of the regression coefficient calculated when either or both the input and the output parameter values are normalized.

- When only input parameters are normalized, we say it is input normalized regression coefficient.
- When only output parameters are normalized, we say it is output normalized regression coefficient.
- When both input and output parameters are normalized, we say it is input-output normalized regression coefficient.

ADE GXL uses the following equations to normalize the data:

- To normalize input values for design variables and statistical variables:

  `Xnorm = 2*(X-Xmin)/(Xmax-Xmin)`

- To normalize input values for Monte Carlo analysis:

  `Xnorm = (X-Xmean)/Xstd`

- To normalize output values for design variables and statistical variables:

  `Ynorm = (Y - Ymean)/Ymean`, if `Ymean` != 0

Normalized regression coefficients allow comparing the results between different specs for the same parameter (output normalized), same specs for different parameters (input normalized), and also between different parameter/spec combinations (input-output normalized).

To view regression data:

1. Choose *Single Variable Linear Regression*, *Single Variable Quadratic Regression*, or *Multi Variate Linear Regression* from the drop-down list at the top of a tab.

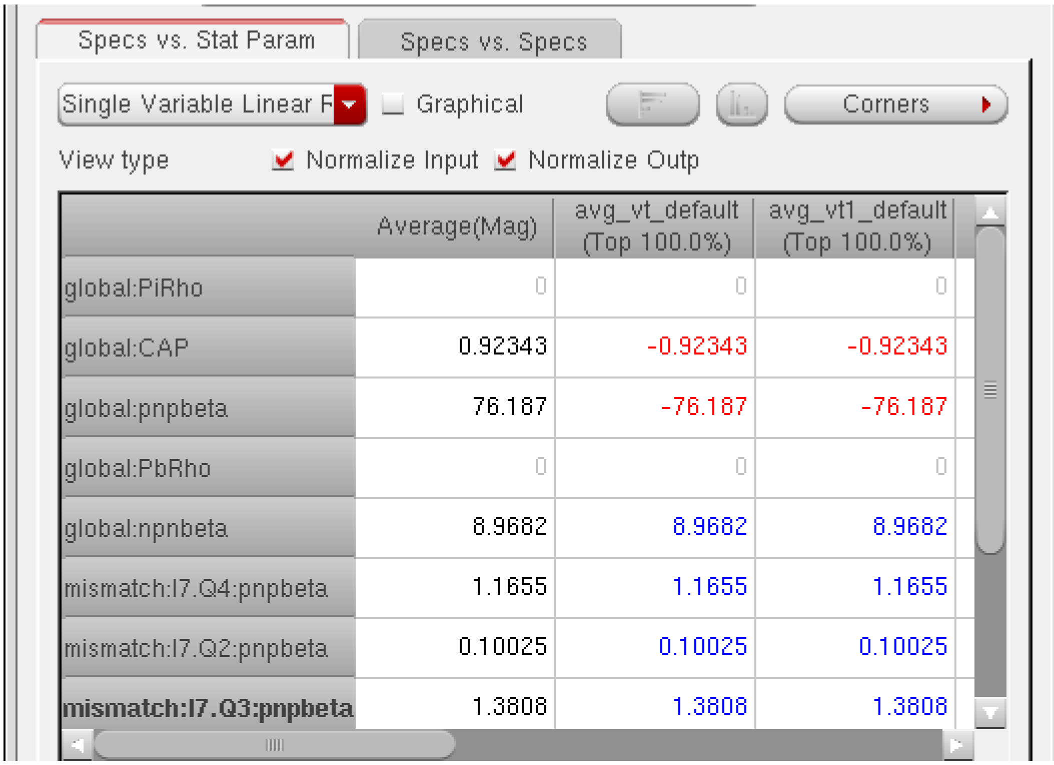
   The regression data appears.

2. Do one of the following:

   - To normalize the input data, select the *Normalize Input* check box.
   - To normalize the output data, select the *Normalize Output* check box.
   - To normalize both the input and the output data, select the *Normalize Input* and *Normalize Output* check boxes.

   The regression data appears.

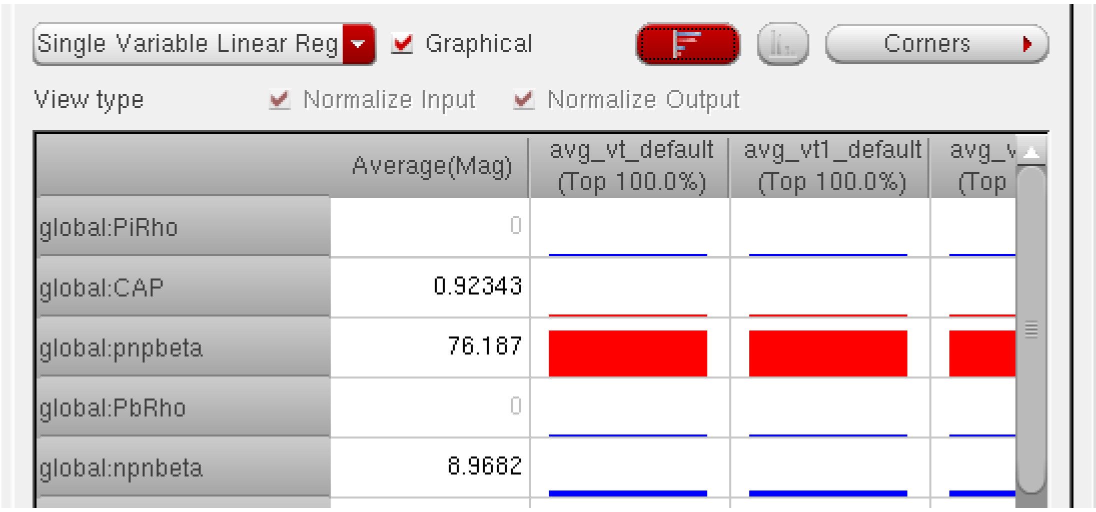
Figure 2-8 Regression Data in Sensitivity Analysis window



The first column lists the variables, and the *Average(Mag)* column displays the average of absolute values of normalized regression coefficients for each specification.

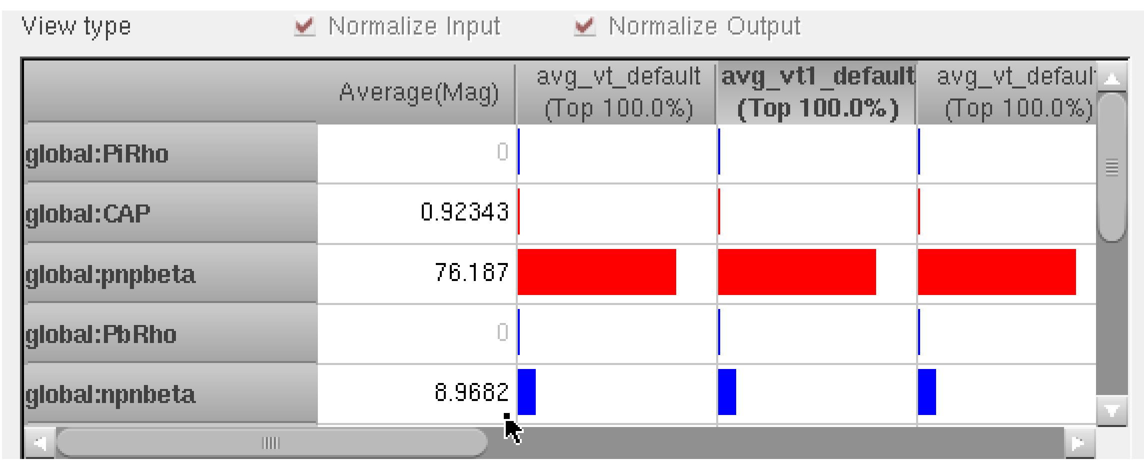
Viewing Regression Data in Graphical Format

In graphical format, the regression data is shown as bars. By default, the data is displayed as vertical bars, as shown in the following figure.



Note: Here, positive regression values are displayed in blue color and negative values are displayed in red color.

To display the regression data as horizontal bars, click the  button next to the *Graphical* checkbox. The graphical data is changed as shown in the following figure.

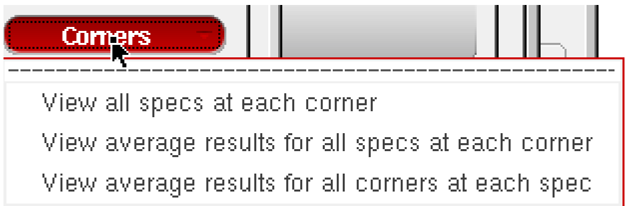


Important

When you view data in graphical format, the *Normalize Input* and *Normalize Output* options are automatically disabled. In addition, these two options are automatically selected when you are viewing single variable or multivariate linear regression data. For correlation data, these two options are not used.

Changing the Data View Type

A view type specifies the data that you want to view in inputs and outputs. You can specify the data view type by using the following list at top right of the tab.



The available view types and their descriptions are listed in the following table.

View Type	Description
View all specs at each corner	View detailed results for all corners and specs. This is the default view.
View average results for all specs at each corner	View average results of all specs at each corner.
View average results for all corners at each spec	View average results at all corners for each spec.

You can plot the average results displayed by the view types listed in the table given above.

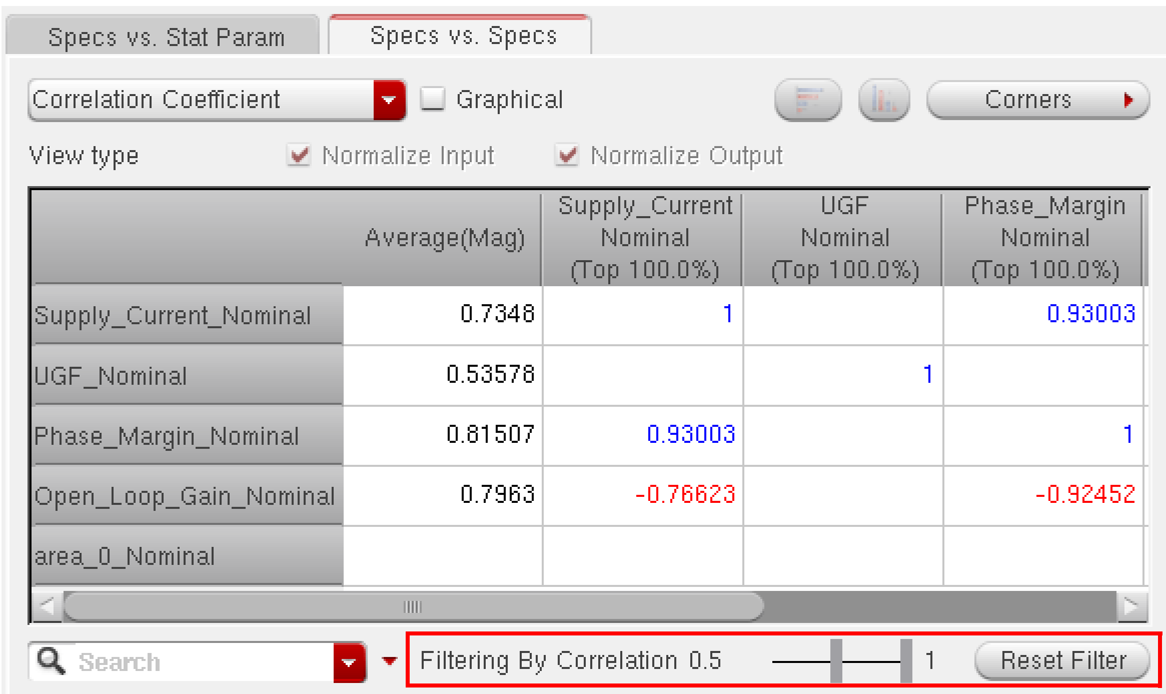
See also:

- Hiding Corner Data on page 72
- Hiding Specification Data on page 72
- Hiding or Showing the Detailed Results for Corners on page 72
- Hiding or Showing the Detailed Results for Specifications on page 73
- Hiding or Showing Statistical Parameters on page 73
- Showing All Hidden Data on page 73

Filtering Data Based on Correlation Coefficient

With the *Filter by Correlation* slider bars, you can choose to filter out relationships that are in a range of two given correlation values. By default, all the results in the range of `0` and `1` are displayed.

You can move the slider bars to specify the correlation filter range. For example, if you move the slider bars as shown below, ADE GXL displays data with correlation coefficient values between 0.5 and 1.0.



Filtering Data Based on Columns

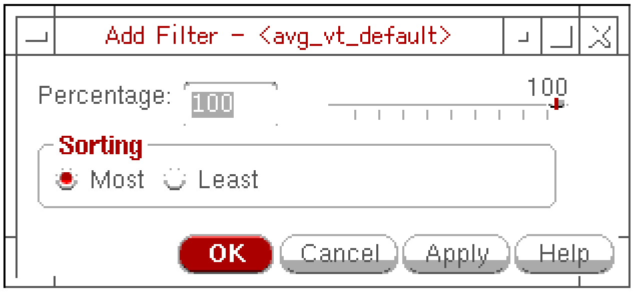
To find out the most or least sensitive parameters, you can apply a filter on a column. This will filter out parameter rows based on the value of the column data. When the correlation table is displayed, the filter is applied to the correlation data. When the regression table is displayed, the filter is applied to the regression data.

To filter data based on a column:

1. Select a column based on which you want to filter the data.
2. Right-click and choose *Add Filter*.

   The Add Filter form is displayed.

Figure 2-9 Add Filter Form

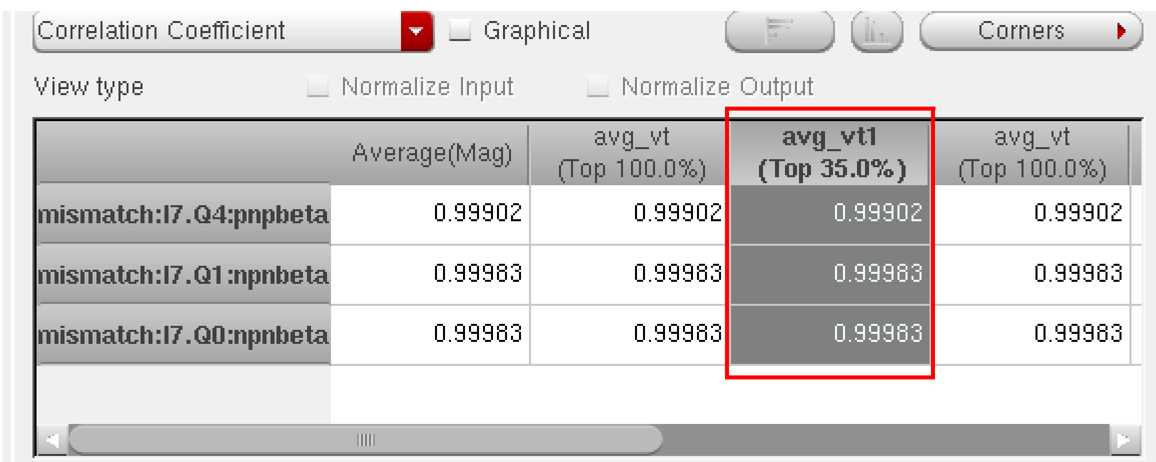


3. Specify a percentage value.

   Alternatively, use the percentage slider to specify a percentage of parameters to be filtered.

4. In the *Sorting* section, select any one of the two options, *Most* or *Least*, to display a specific percentage of top or bottom rows according to the sensitivity of parameters.

   For example, if you specify `35` as the percentage value and `Most` as the sorting criteria, ADE GXL will display the top 35% of rows that show strong correlation, as shown in the following figure.



Note: You can apply separate filters on multiple columns. In that case, ADE GXL filters data from the entire row set for each column condition separately and then displays the row intersection (rows that are common in all resulting row sets).

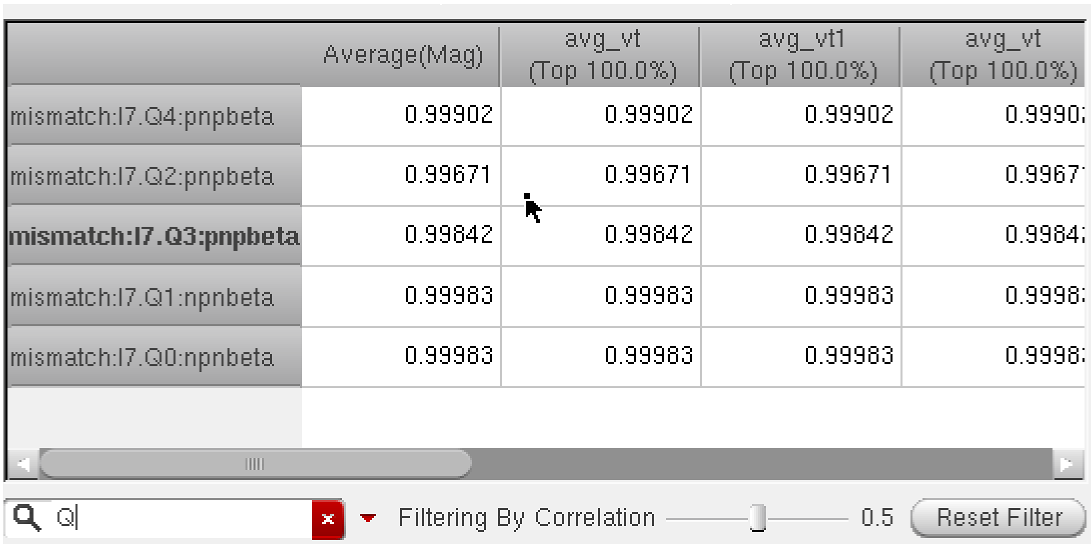
To remove the filters on columns:

➡ Right-click on a column and choose *Remove Filter*.

➡ Click *Reset Filter* on the Sensitivity Analysis window. This removes all filters applied on the form.

Filtering Data Based on Search Criteria

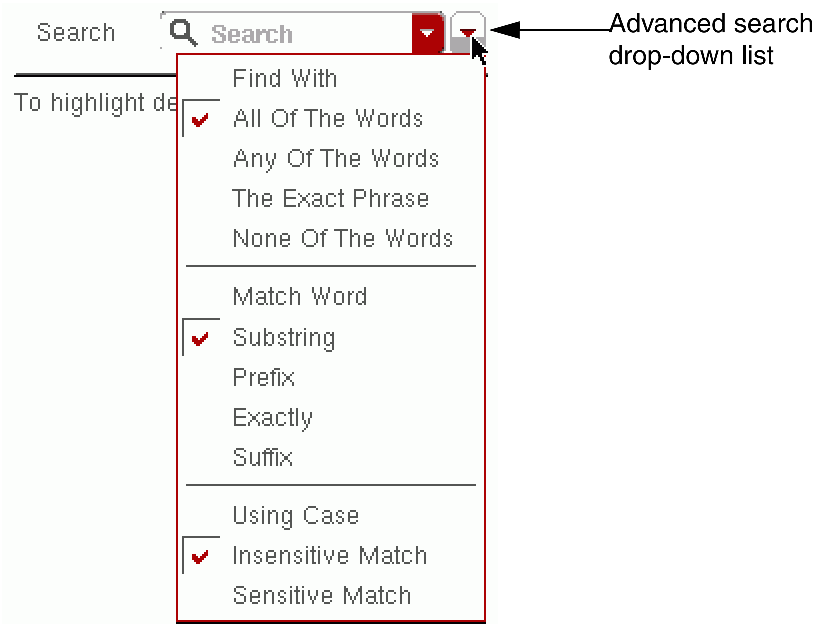
The *Search* field allows you to filter the results in the Sensitivity Analysis window to view only the results for specific parameters. For example, if you type a string `Q` in the *Search* field, the form updates automatically to show only the results for the parameters whose names start with the string `Q`, as shown in the following figure.



To clear the search, click the button in the *Search* field.

Previous filters used in the current session are accessible from the drop-down list in the Search field.

The *Advanced* search drop-down list next to the *Search* field provides commonly used search options to refine your search results.



For more information on the *Search* field, see <u>The Search Toolbar</u> in the *Virtuoso Schematic Editor XL User Guide*.

Resetting Filters

To reset all the filters applied on sensitivity results, click *Reset Filter* next to the *Filter by Correlation* slider bar.

ADE GXL removes all the applied filters and displays all the data for that view.

Sorting Data

To sort data in a row or a column in the Sensitivity Analysis window:

➡ Right-click on a row or column heading and choose any of the following commands, as required:

- ❑ *Sort by value*: This command sorts all the data based on the value in each cell.
- ❑ *Sort by absolute value*: This command sorts all the data based on the absolute value in each cell in the selected column or row.

Data is always sorted based on normalized sensitivity regression coefficients.

Hiding Corner Data

You can do the following to hide corner data:

- To hide the data for a specific corner, right-click on the column for the corner and choose *Hide Corners*.
- To hide the data for all corners other than a specific corner, right-click on the column for the corner you don't want to hide and choose *Hide Other Corners*.

To show all hidden data, right-click and choose *Show All*.

Hiding Specification Data

You can do the following to hide specification data:

- To hide the data for a specification, right-click on the column for the specification and choose *Hide Specs*.
- To hide the data for all specifications other than a specific specification, right-click on the column for the specification you don't want to hide and choose *Hide Other Specs*.

To show all hidden data, right-click and choose *Show All*.

Hiding or Showing the Detailed Results for Corners

When you are viewing average results for all specs at each corner, you can do the following to hide the detailed results for corners:

- To hide the detailed results for a corner, right-click on the column for the corner and choose *Collapse Selected Corners*.
- To hide the detailed results for all corners, right-click and choose *Collapse All Corners*.

You can do the following to show the detailed results for corners in the Corner View:

- To show the detailed results for a corner, right-click on the column for the corner and choose *Expand Selected Corners*.
- To show the detailed results for all corners, right-click and choose *Expand All Corners*.

Hiding or Showing the Detailed Results for Specifications

When you are viewing average results for all corners at each spec, you can do the following to hide the detailed results for specifications:

- To hide the detailed results for a specification, right-click on the column for the specification and choose *Collapse Selected Specs*.
- To hide the detailed results for all specifications, right-click and choose *Collapse All Specs*.

You can do the following to show the detailed results for specifications in the Spec view:

- To show the detailed results for a specification, right-click on the column for the specification and choose *Expand Selected Specs*.
- To show the detailed results for all specifications, right-click and choose *Expand All Specs*.

Hiding or Showing Statistical Parameters

Statistical parameters are displayed in the Specs vs. Parameter tab and the DC Ops vs. Parameter tab if the *Enable variation of Statistical Parameters* check box is selected in the Sensitivity form.

To hide the statistical parameters, right-click and choose *Hide Statistical Parameters*.

To show all hidden data, right-click and choose *Show All*.

Showing All Hidden Data

To show all hidden data, right-click and choose *Show All*.

Highlighting Associated Devices in the Schematic

When you right-click on parameters in the Sensitivity Analysis window and choose *Highlight on Schematic*, the associated device(s) in the schematic are highlighted.

Note: For Monte Carlo runs, you can highlight only the devices associated with mismatch parameters.

Viewing Results in Multiple Sensitivity Analysis Windows

If you want to view the results of Sensitivity Analysis in different view modes and with different filtering conditions at the same time, you can open the same results in multiple windows. Next, you can apply different view settings and filters in different windows and compare the results.

To open the results in multiple windows, perform the following steps:

1. In the *Data View*, right-click on a history run and choose *Sensitivity Results*.

   A new Sensitivity Analysis window is opened with the results of the selected history displayed in it.

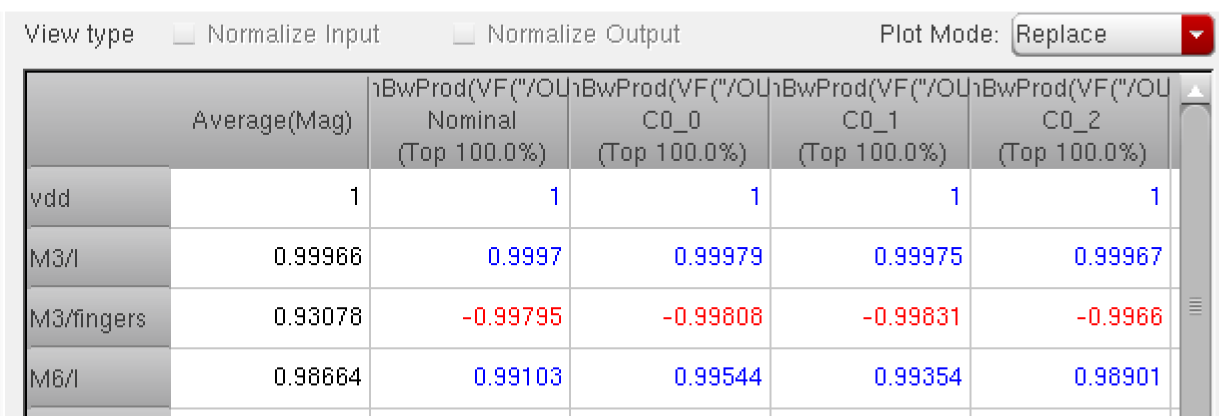
2. If required, customize the view of results as per your requirements.

3. In the Sensitivity Analysis window, choose *File — New Window*.

   A new Sensitivity Analysis window is opened and the same results are displayed in it. The data view settings in the new window are same as those in the first window. However, you can customize these, as required. Now, the same results are visible in two different windows with different view settings.

Changing Number Format

By default, in numeral format, the number of significant figures used to display data is not fixed. In the results shown below, the data appears in different significant digits.



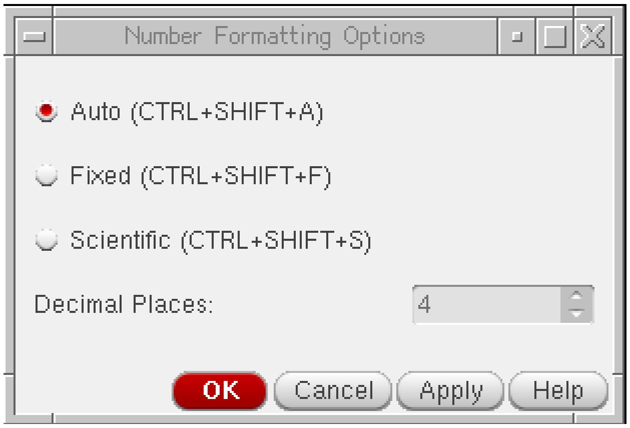
	Average(Mag)	ıBwProd(VF("/OU Nominal (Top 100.0%)	ıBwProd(VF("/OU C0_0 (Top 100.0%)	ıBwProd(VF("/OU C0_1 (Top 100.0%)	ıBwProd(VF("/OU C0_2 (Top 100.0%)
vdd	1	1	1	1	1
M3/l	0.99966	0.9997	0.99979	0.99975	0.99967
M3/fingers	0.93078	-0.99795	-0.99808	-0.99831	-0.9966
M6/l	0.98664	0.99103	0.99544	0.99354	0.98901

If required, you can set a fixed number of significant digits in which you want to display the numeral data. For this, you can use the **Number Formatting Options** form.

To change the number format:

1. In the Sensitivity Analysis window, choose *Format — Numbers*.

The **Number Formatting Options** form is displayed. Note that the default format is *Auto*.



2. Select *Fixed* to display the numbers in a fixed point format.

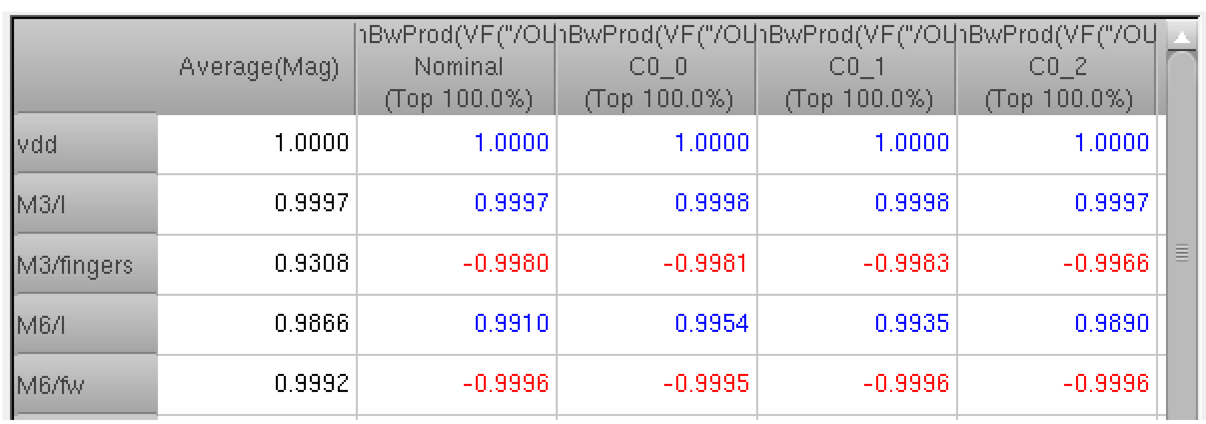
   Note: Similarly, you can select *Scientific* to display the numbers in scientific format.

3. In the *Decimal Places* field, specify the number of decimal places that you want to show for each number in the results.

   Note: Decimal places can be set only upto a maximum of six digits.

4. Click *Apply* to apply the changed format.

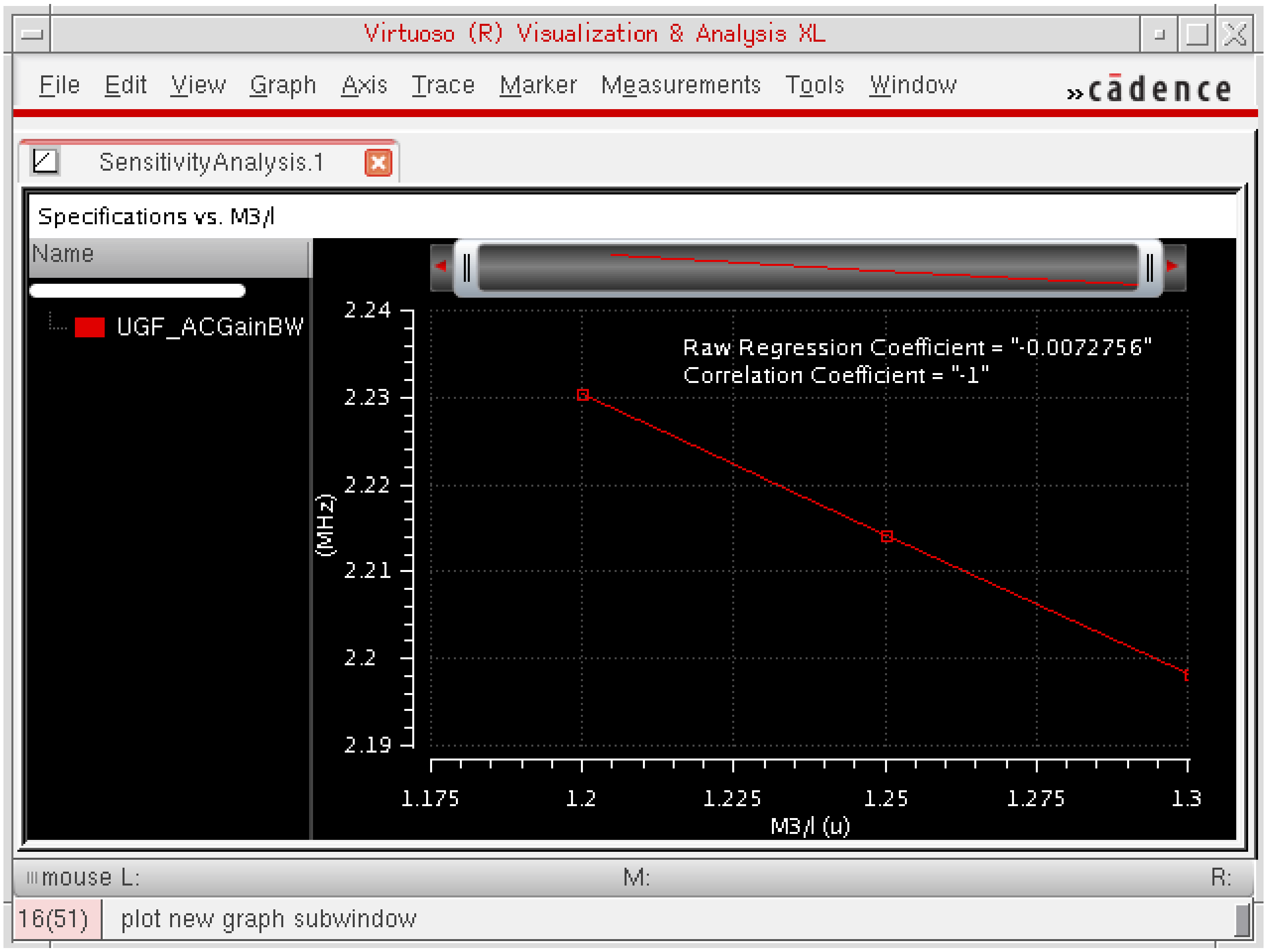
In the results shown below, the data appears in the fixed format with four decimal places.



Plotting Sensitivity Analysis Results

The sensitivity analysis results can be plotted in the Virtuoso Visualization and Analysis XL window. The data can be plotted in three ways:

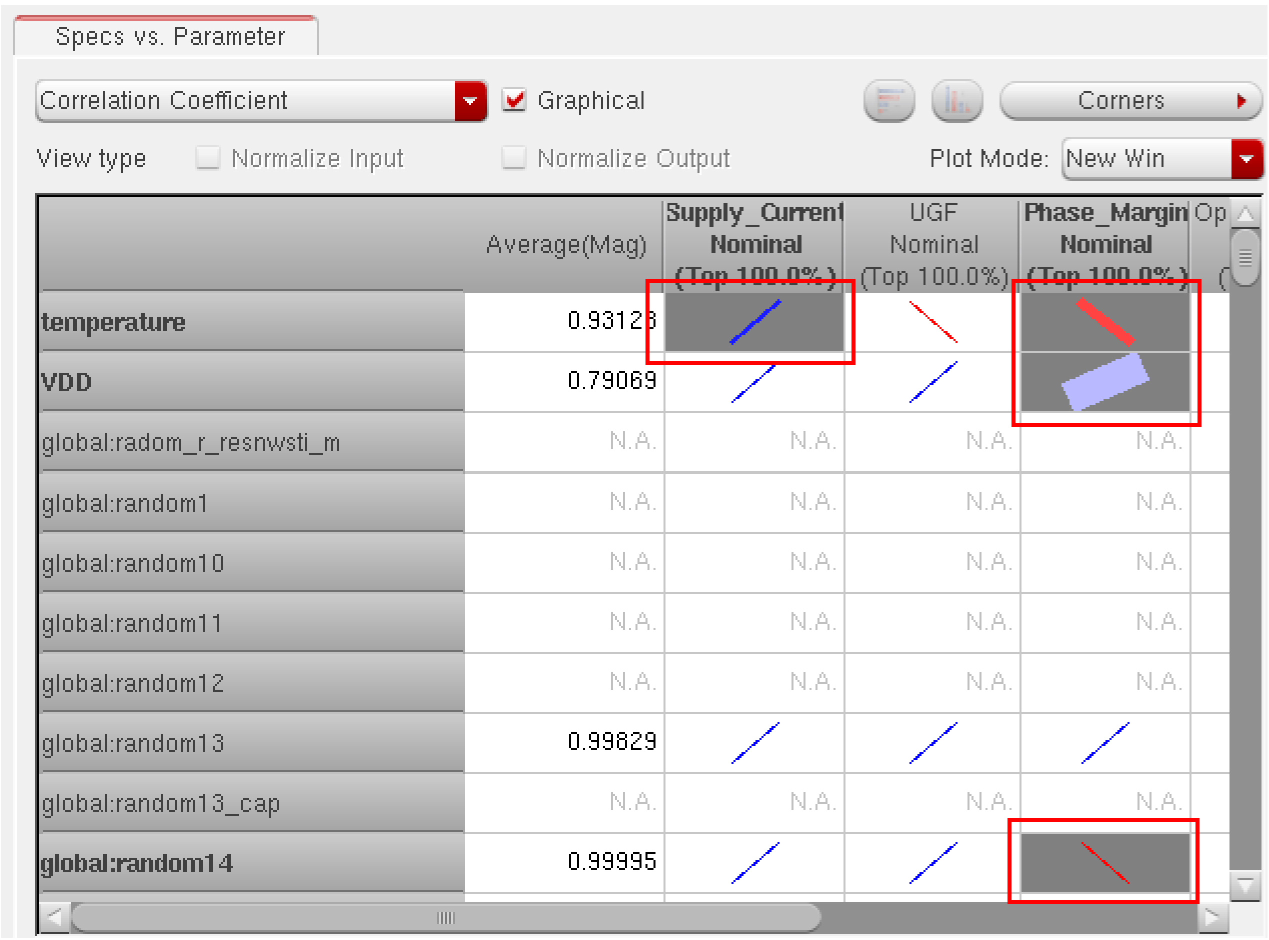
➡ Double-click any data cell in the Sensitivity Analysis results window. The correlation and regression data in the plotted graph exhibits the sensitivity of the selected specification to the corresponding parameter, as shown below.



■ Select two or more parameter cells for a single spec column, right-click, and choose the *Plot Across Selected Parameters* command. This displays the sensitivity of multiple selected parameters to the specification represented by the corresponding column, as shown in the following figure:

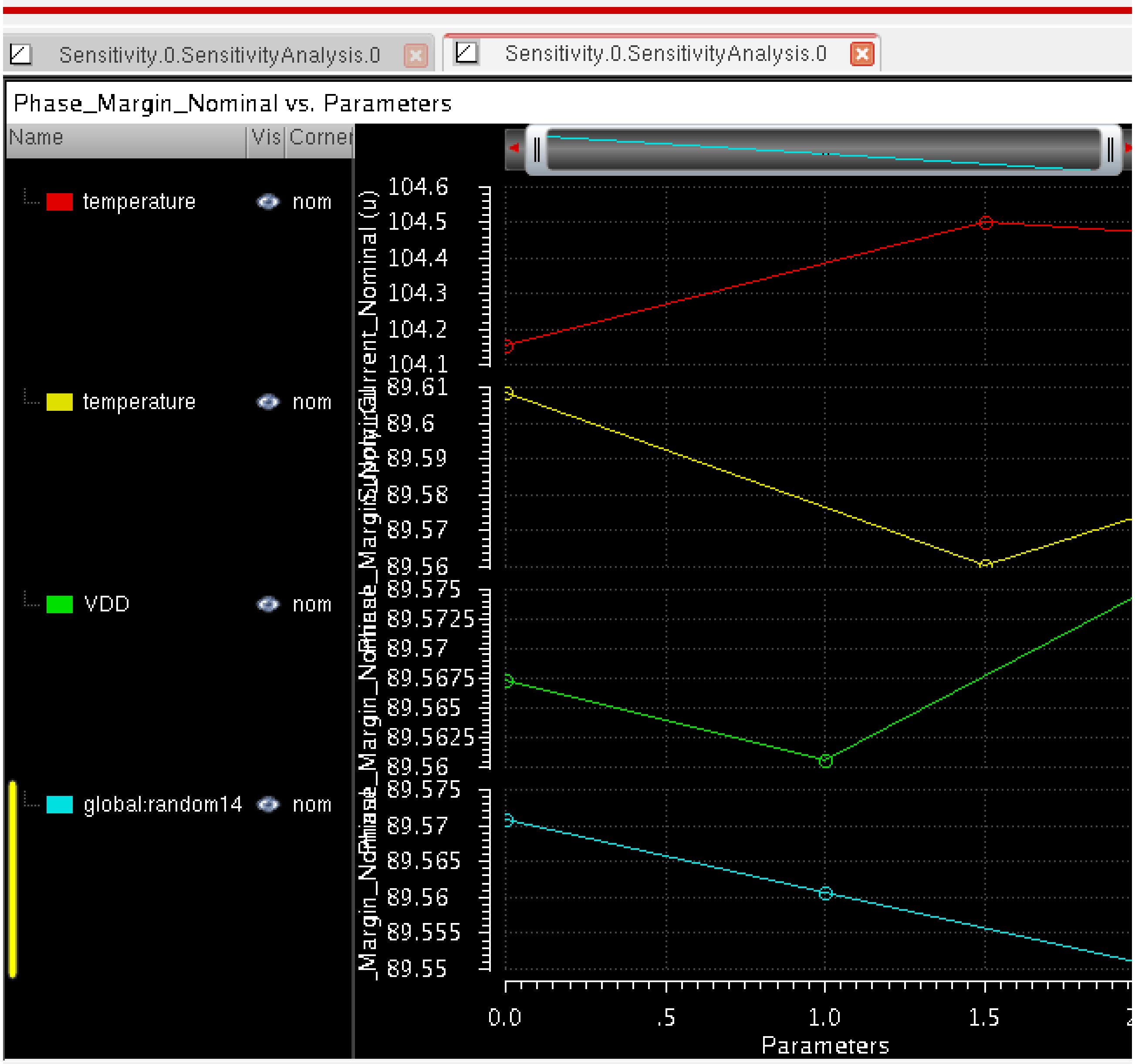
➡ Select two or more data cells in different spec columns, right-click, and choose the *Plot Across Parameters* command to display the sensitivity of the selected parameters to the selected specifications.

For example, select cells for different specs and parameters, as shown in the following figure:



Now, right-click and choose *Plot Across Parameters.*

In this case, the result for each parameter and spec combination is plotted in a separate strip, as shown in the figure below.



Tip

A graph is by default plotted as a continuous line. However, you can choose to plot a scatter plot by setting the `sensitivityPlotContinuousLine` environment variable in the `.cdsenv` file or in CIW as shown below.

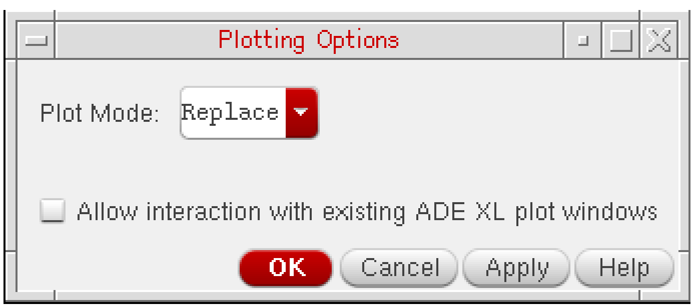
```
envSetVal("adexl.plotting" "sensitivityPlotContinuousLine" 'boolean nil)
```

If the Virtuoso Visualization and Analysis XL window is already open for the selected history, by default, a new plot always replaces the previous plot. However, you can specify different plotting modes depending on your data analysis requirements. For more details, refer to the following sections:

- Setting the Plotting Mode
- Displaying the ADE XL and Sensitivity Analysis Results in the Same Graph
- Plotting Average Sensitivity Analysis Results
- Displaying Spec Markers in Sensitivity Analysis Plots

Setting the Plotting Mode

You can set the plotting mode for the sensitivity analysis results graph by using the Plotting Options form.



In the Sensitivity Analysis Results window, choose *Options – Plotting*. The Plotting Options form is displayed. In this form, the *Plot Mode* field specifies the plotting mode of the graph. By default, the default plotting mode is set to `Replace`.

You can select any one of the following plotting modes:

- Replace
- Append
- New SubWin
- New Win

Tip

You can also set the plot mode by using the *Plot Mode* drop-down list on the Sensitivity Analysis results window.

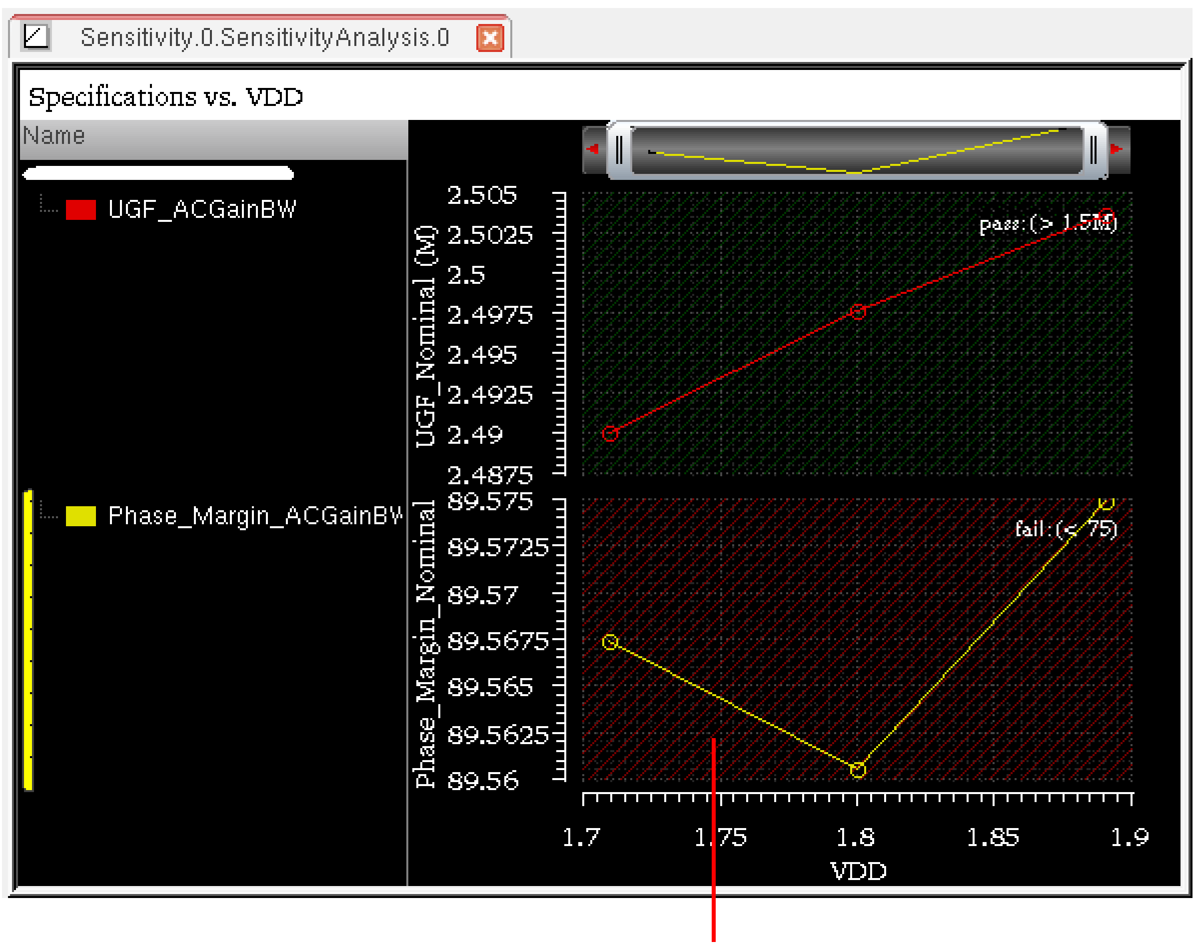
Replace Plot Mode

In this mode, the new plot always replaces an existing graph, if any. Otherwise, a new window appears that displays the graph.

Append Plot Mode

In this mode, if a graph is already open for the selected history, the new plot is appended to the existing graph. Otherwise, the new plot is displayed in a new window.

Consider an example. A graph is already open for the selected history and displays the sensitivity of `Open_Loop_Gain` to the parameter `M12/I`. Next, if you set the plot mode to *Append* and plot the sensitivity of `Offset_Voltage` to the parameter `M12/I`, the new plot is appended to the existing graph, as shown below.



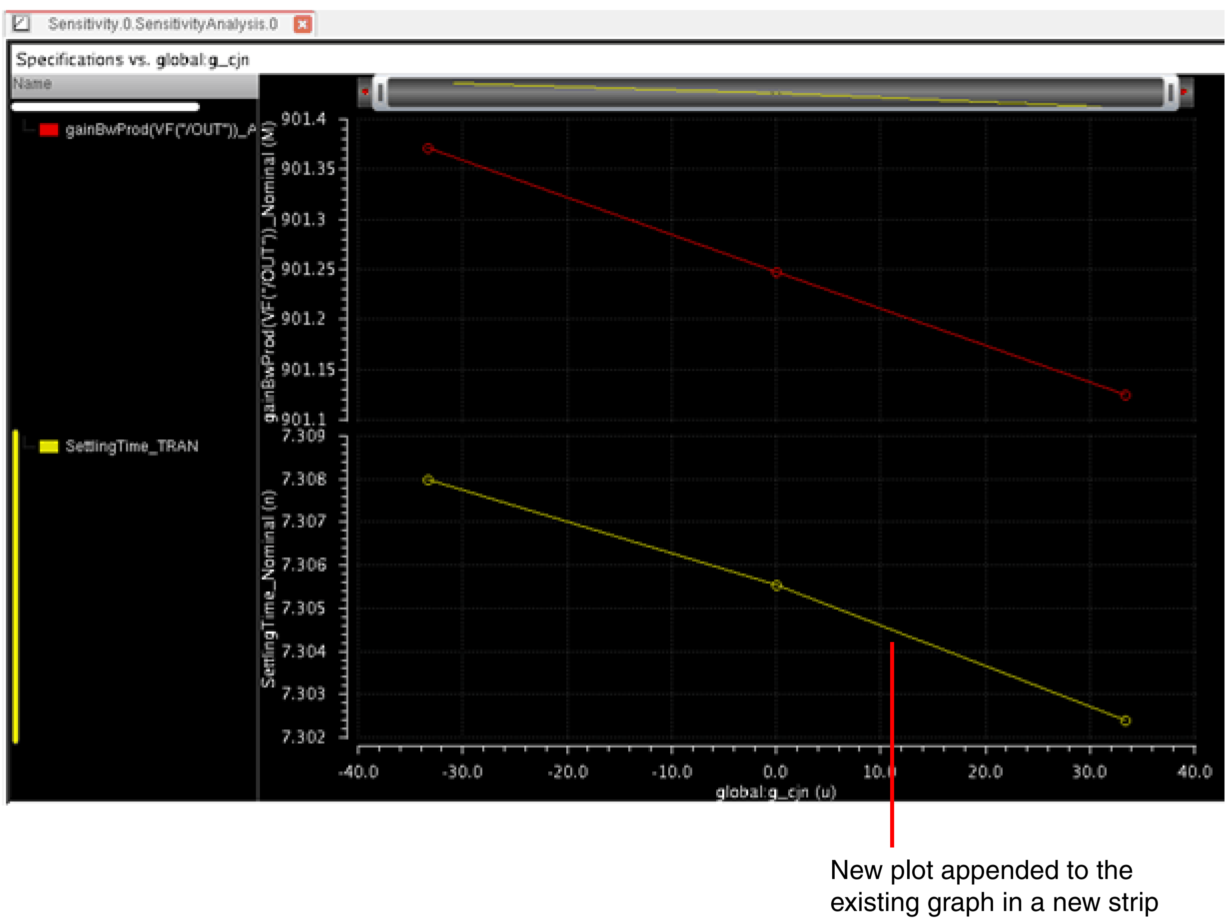
New plot appended to the existing graph, in a new strip

Note: For the plot to append to the existing graph, it is important that you plot another graph for the same parameter. ADE GXL does not retain plots with different x axes on the same graph. In the example shown above, if you plot the results for a different parameter, the graph will be plotted in a new window. However, if you select multiple results for different parameters and choose *Plot Across Parameters*, all the plots are appended to the same graph and the x axis will show `Parameters`.

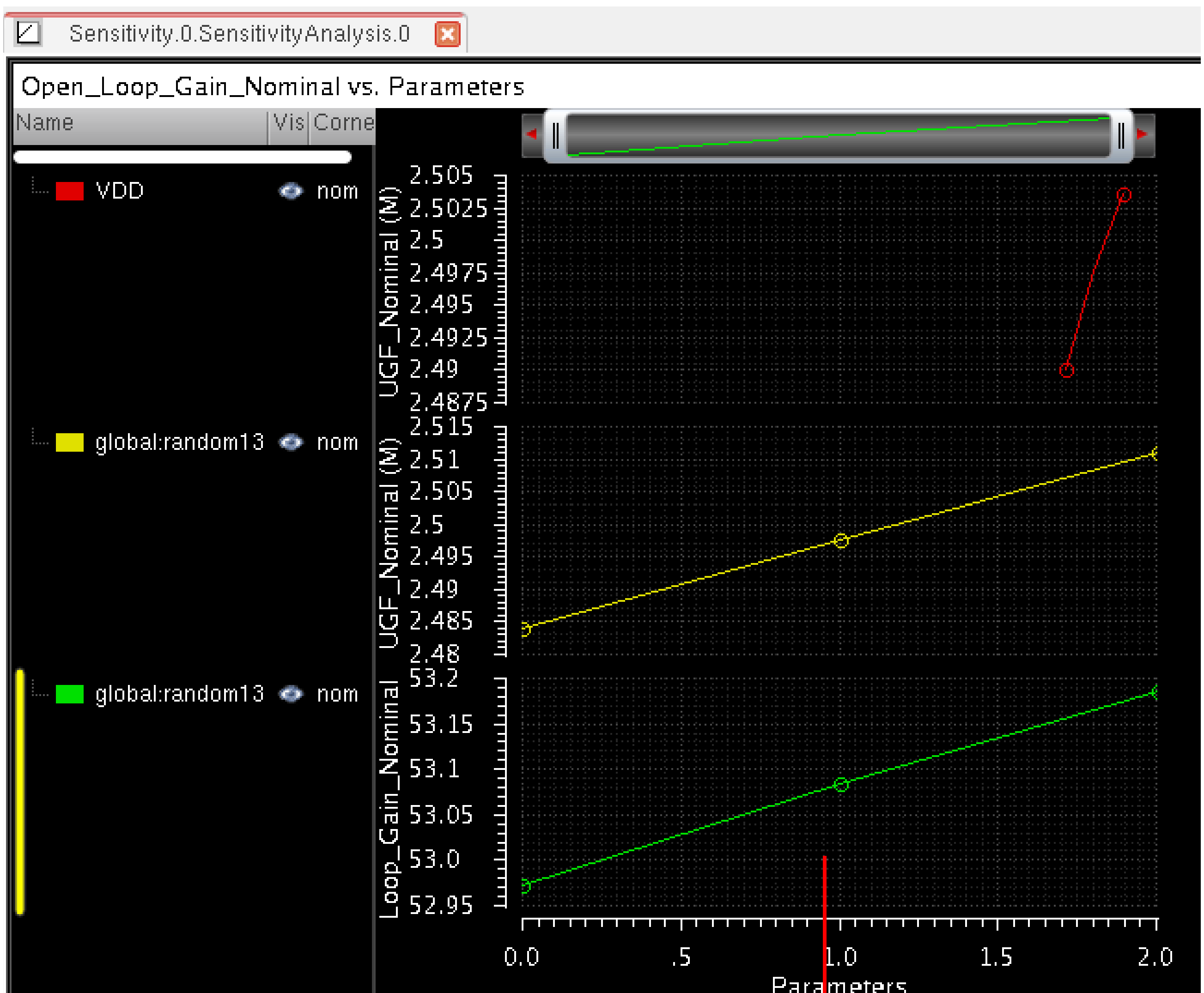
Exceptions to the Append Plot Mode

When the Plot Mode is set to Append, in the following two cases, the new plot is displayed in a new strip instead of being appended to the existing graph:

- If the new plot is for the same parameter but a different specification as that of the graph plotted earlier, the new plot is displayed in a new strip instead of being appended to the existing graph, as shown below.



- If you plot multiple parameters for the same specifications, and next, plot the results for new parameters, all the plots are moved to separate strips. For example, first, you plot `UGF` vs. `VDD` and `UGF` vs. `random13`. The plots are appended to the same graph because the specification `UGF` is common. Next, if you plot the results for another specification `Loop_Gain` vs. parameter `random13` by using the *Plot Across Parameters* command, all the plots will move to separate strips in the existing graph, as shown below.



Plots moved to separate strips
in the existing graph

Note: When plotting is done across different parameters, the data is normalized by using the following equations:

- For design variables and statistical variables:

  `Xnorm = 2*(X-Xmin)/(Xmax-Xmin)`

- For Monte Carlo analysis:

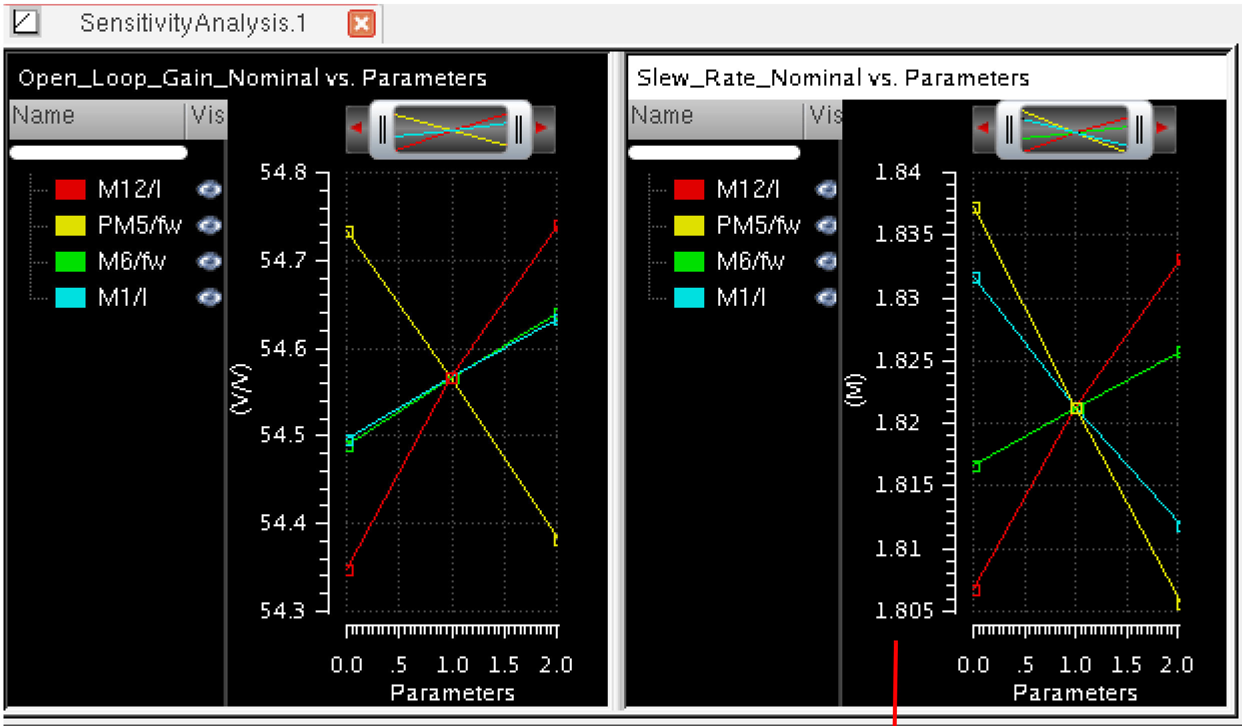
  `Xnorm = (X-Xmean)/Xstd`

New SubWin Plot Mode

In this plot mode, if a graph is already open for the selected history, the new plot is added to a new subwindow in the existing graph.

Consider an example. You have plotted the sensitivity data for the `Open_Loop_Gain` specification to parameters `M12/I`, `PM5/fw`, `M6/fw`, and `M1/I`.

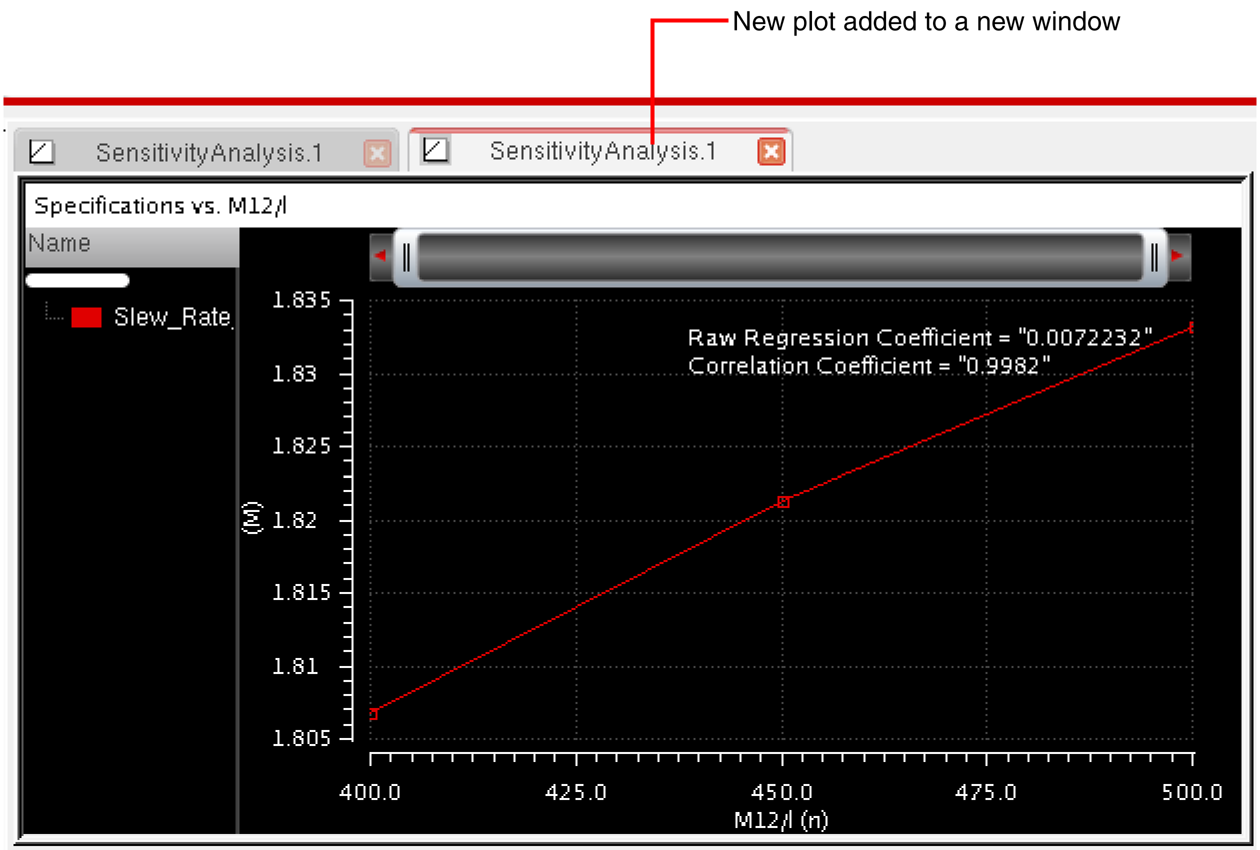
Next, you want to plot and compare the sensitivity data for `Slew_Rate` to the same set of parameters. If you set the plot mode to `New SubWin` and plot data for these parameters, the graph appears as shown below.



New plots added to a new subwindow

New Win Plot Mode

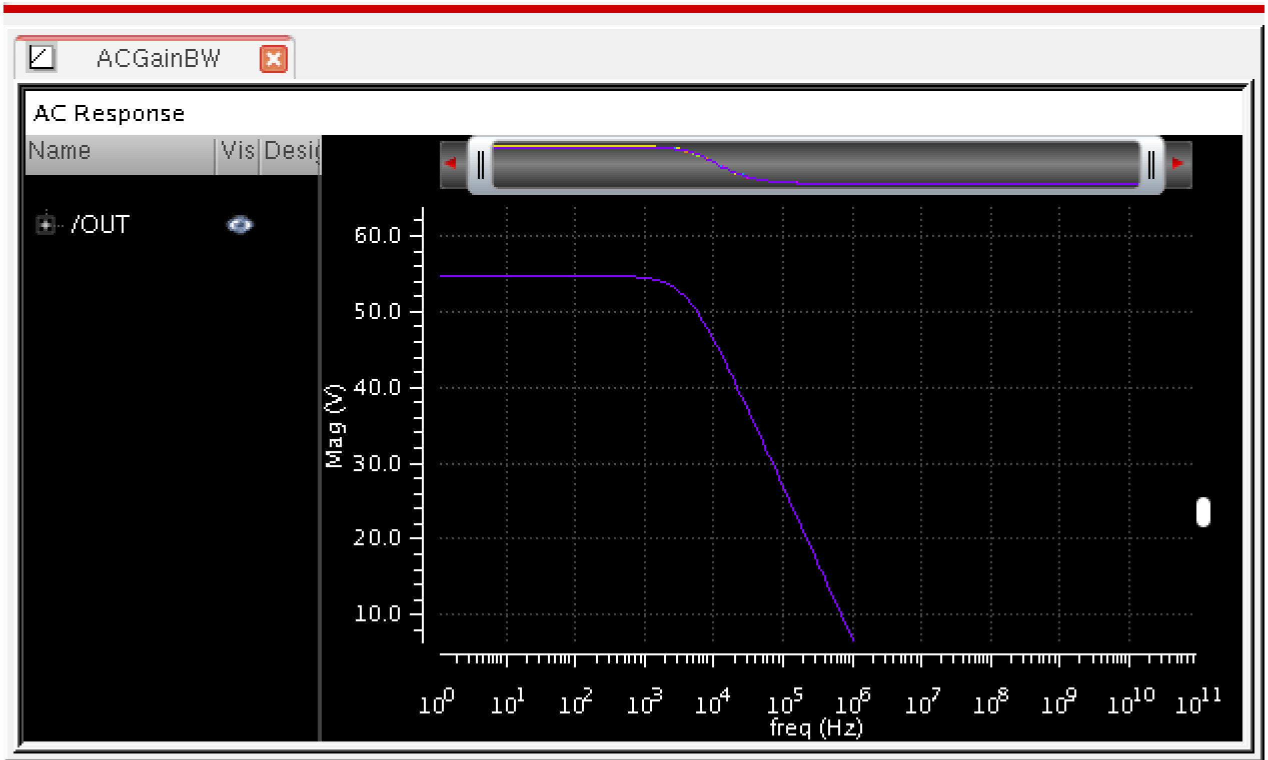
In this plot mode, for every new plot, a window is added to the existing Virtuoso Visualization and Analysis XL window, as shown below.



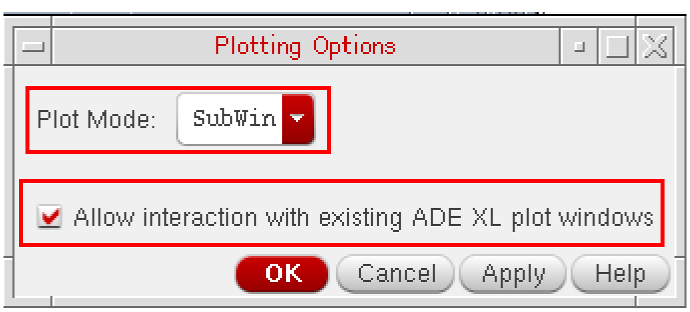
Displaying the ADE XL and Sensitivity Analysis Results in the Same Graph

If you have already plotted results from the ADE XL window, you can choose to plot the sensitivity analysis results in the same graph. To do this, select the *Allow interaction with existing ADE XL plot windows* check box in the Plotting Options form.

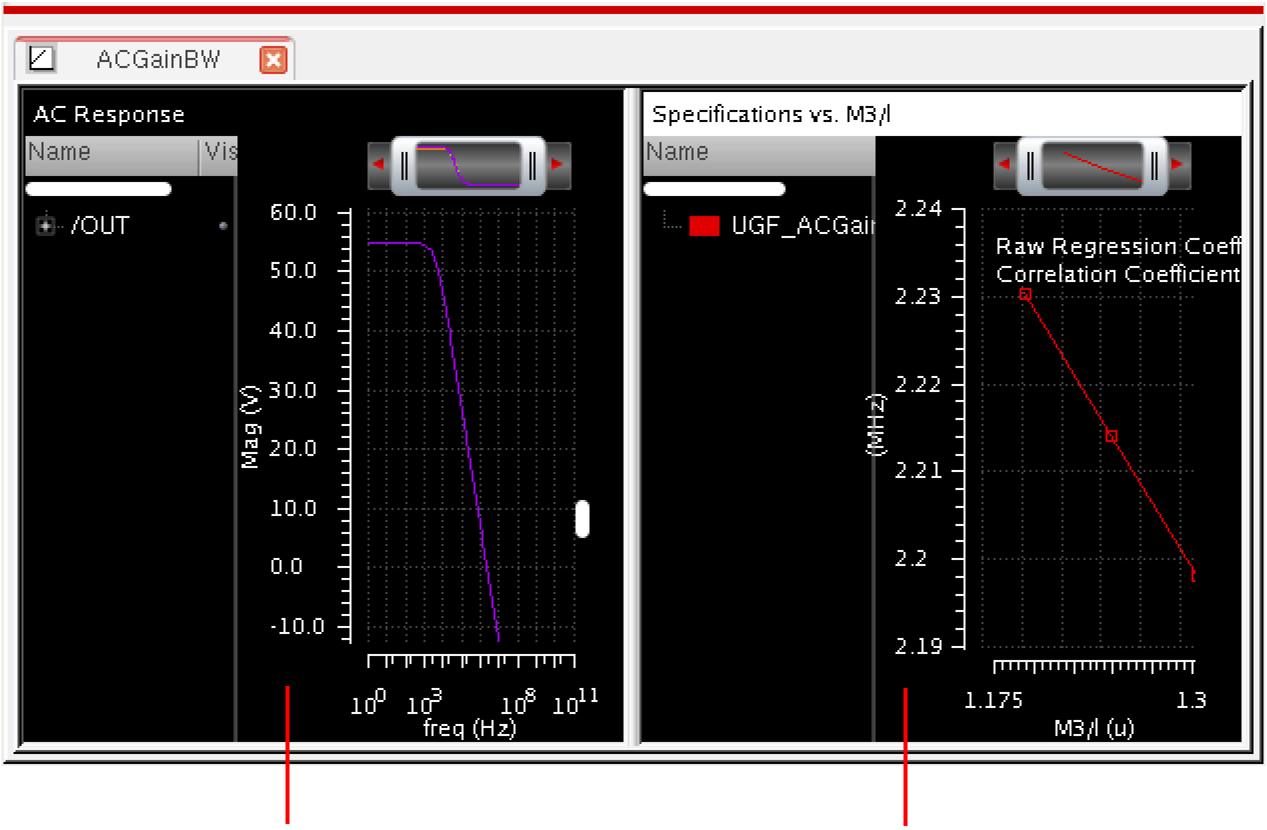
For example, when you plot an output, `ACGainBW`, from the ADE XL results tab, the graph appears as shown below.



Now, if you want to plot the sensitivity results in a new subwindow of this graph to show sensitivity of `UGF_ACGainBW` to the variable `M3/I`, set the plotting options as shown below:



Now, if you plot the sensitivity results, a new subwindow is added as shown below.



Graph plotted from the ADE XL Results tab

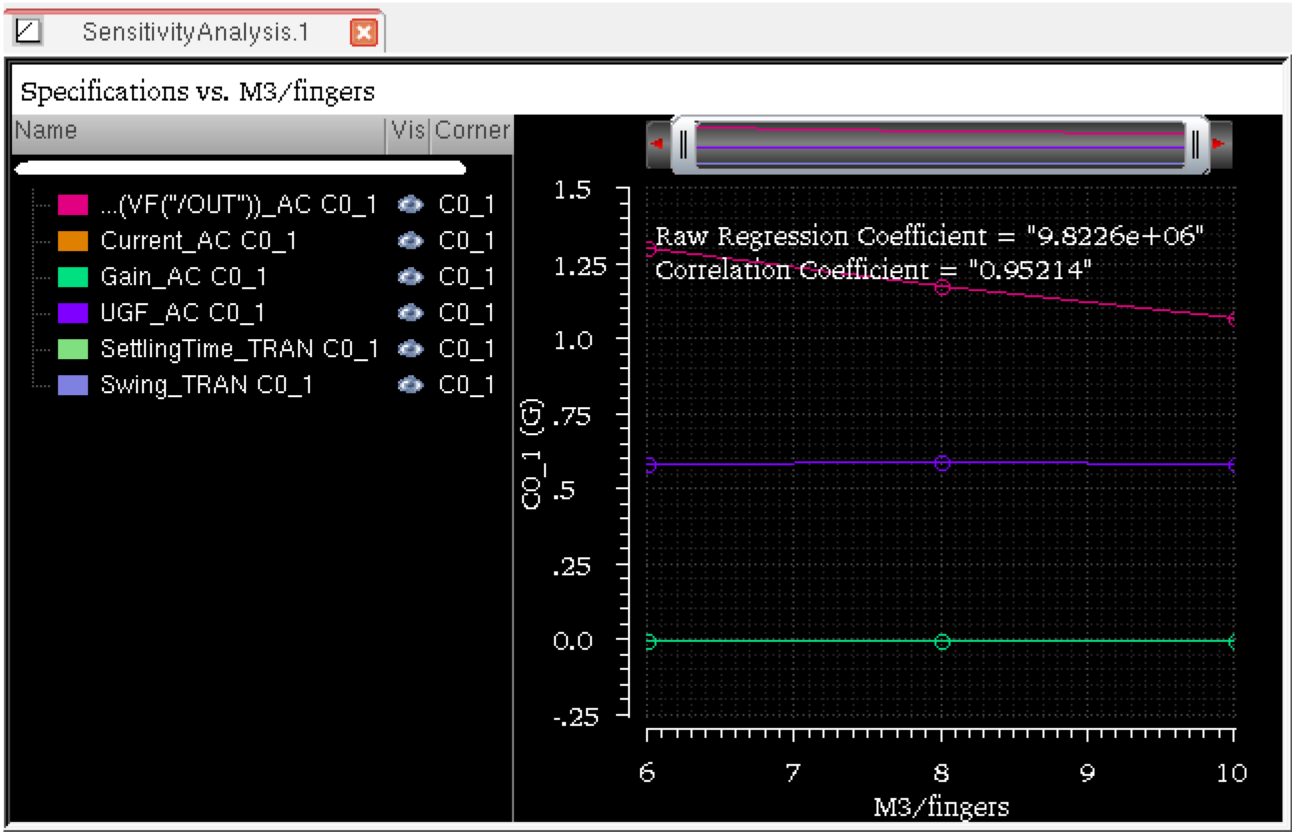
New subwindow added to the same graph to display sensitivity results

Note: The *Allow interaction with existing ADE XL plot windows* check box is not considered when the plot mode is set as *Append*. If this check box is selected and the *Plot Mode* is set to *Append*, the plots for the sensitivity data are not appended to the existing graph. Instead, the sensitivity data is plotted in a new window.

Plotting Average Sensitivity Analysis Results

If you change the default Sensitivity Analysis results view and display average results of all the corners for each spec or results of all specs for each corner, you can double-click in a cell to show all the plots.

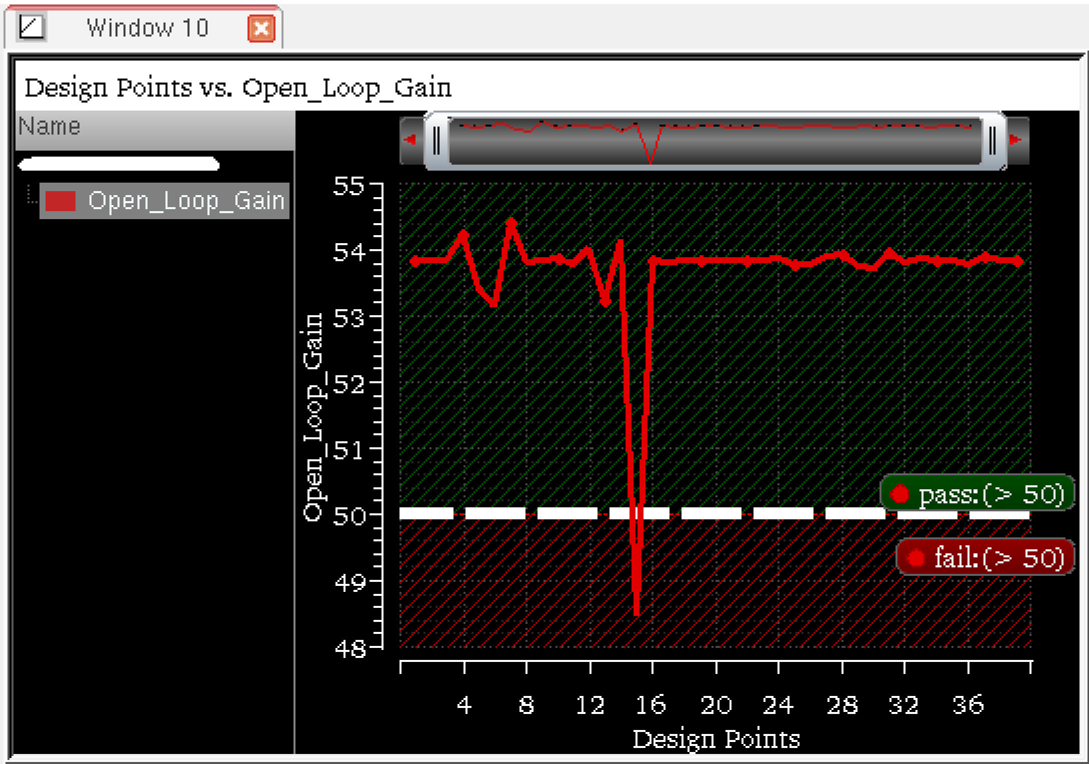
For example, if you display average results of all corners at each spec, when you double-click in a cell, the plots are displayed as shown below.



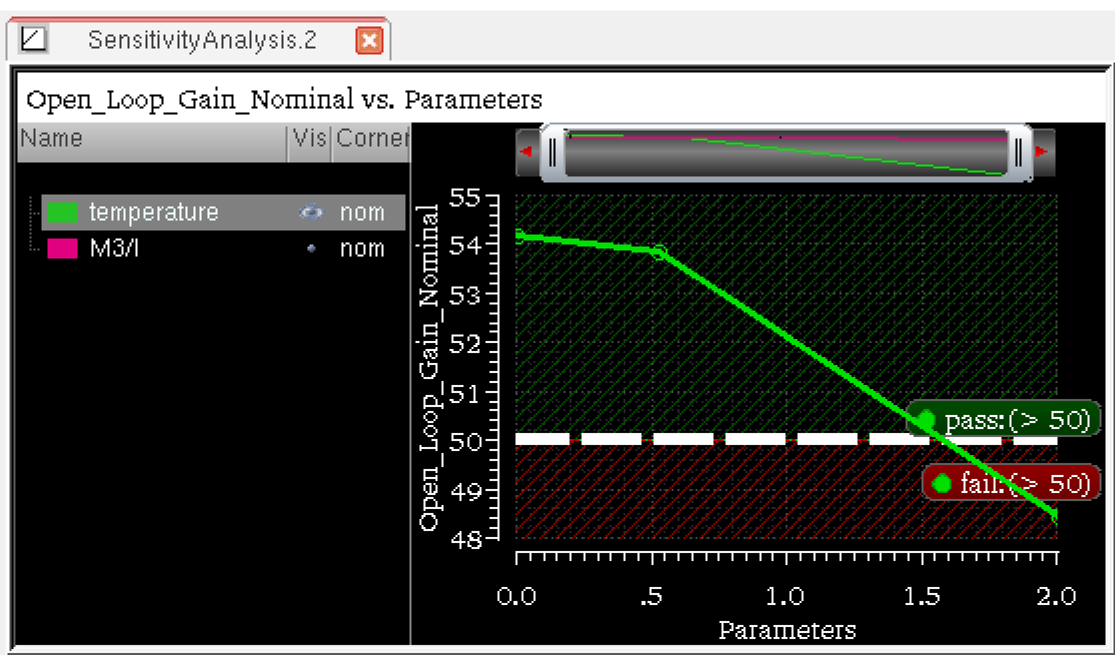
Similarly, you can plot average results of all corners at each spec.

Displaying Spec Markers in Sensitivity Analysis Plots

Similar to the ADE XL graphs, you can display spec markers in the graphs plotted for Sensitivity Analysis results. For this, on the Results tab of ADE XL, choose *Options —Plotting/Printing*. On the **ADE XL Plotting/Printing Options** form, select *Spec Markers*. When this option is selected, the graphs displayed from the Results tab of the Output pane show spec markers, as shown below.



The specs vs. parameter graphs displayed from the Sensitivity Analysis window also show spec markers, as shown below.



Saving the Sensitivity Data

You can save the sensitivity data to a text file:

1. Choose *File – Export Results to CSV*, then do one of the following:
    - ❑ Choose *Current Tab* to save only the data in the current tab.
    - ❑ Choose *All Tabs* to save the data in all the tabs.
2. In the Export Results form, specify a path and file name.
3. Click *OK*.

3

Worst Case Corners

A corner is a combination of variables or process models that define a scenario in which you want to measure the performance of your design. For more information about corners, see Simulating Corners in the *Virtuoso Analog Design Environment XL User Guide*.

When you work with large simulation data, defining corner conditions results in hundreds or thousands of possible corner combinations to run. Therefore, running simulations over all corners becomes very time consuming process and also requires large computation resources. Therefore, Worst Case Corners helps to reduce the overall project time devoted to simulation and still provide a high level of accuracy.

By identifying Worst Case Corners, the problem is reduced to a subset of worst case conditions. These worst case corners can then be used in verification and design. In addition, running simulations over a reduced set of corners facilitates design changes to be validated quickly.

This chapter covers the following topics to describe how to use the Worst Case Corner analysis in ADE GXL:

Prerequisites for Setting Up the Worst Case Analysis

Before you create worst case corners, perform the following steps:

1. On the *Run* toolbar, in the *Select a Run Mode* drop-down list, select *Worst Case Corners.*

2. If you want to vary variables, ensure that you have specified variables in the Variables and Parameters assistant.

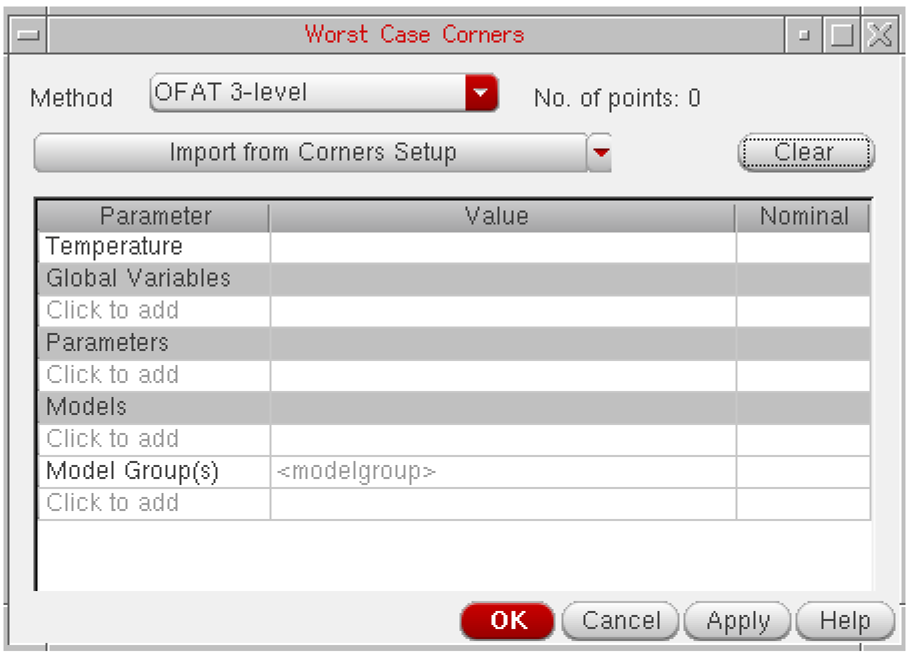
3. On the *Outputs Setup* tab, select the required spec and ensure that for each spec, you specify an objective type and a target value.

   Note: If you specify `tolerance (tol)` or `range` objective types for the spec you choose, two worst case corners are created. For all other objective types, one worst case corner is created.

Specifying the Worst Case Corner Analysis Setup

After you select the run mode, click the *Simulation Options* button to specify the worst case corner setup.

The Worst Case Corners form appears as shown in the figure below.



In the Worst Case Corners form, specify the settings that you want to use while running the sensitivity analysis to identify the worst case corners. In this form, you can specify the Temperature, variables, and models that you want to vary and also specify the method by which you want to vary them.

The various form fields are explained in detail in the following sections:

- Choosing a Method on page 95

Choosing a Method

In the *Method* field, specify the method (algorithm) by which you want to vary the variables. Depending upon the method you specify in this field, the tool, decides how many and which combination of variables are to be simulated.

Following are the five available methods:

- `OFAT 3-level` - In this method, the tool varies each factor for three values. For example, the values of temperature can be specified as `-40`, `27`, `85` with the nominal value equal to `27`. If, instead of three points, you specify a range of values, the tool chooses the following three values of the variable for simulation:
  - minimum value in the range
  - nominal value
  - maximum value in the range

  For example, if the Temperature range is `-40`:`10`:`120` (`-40` in steps of `10` degrees to maximum value of `120`) and the nominal value is `27`, the three values used by the tool are: `-40`, `27`, `120`.

- `OFAT Sweep` - In this method, the tool varies each factor for the specified sweep values. You can either specify sweep values as a range in the `min:step:max` syntax or as a set of space-separated values.

- `Central Composite Design` - This method is similar to OFAT 3-level, but more accurate in the case of interactions among the corner variables. In addition to the points simulated by OFAT 3-level, Central Composite Design simulates the max and min values together for every pair of variables. This method does not simulate all model section values and simulates only first, nominal, and last values. Choose a different method if you want to take all model sections into account.

- `2^K Factorial` - This method simulates all possible combinations of the min and max variable values. Use this method to capture interactions of variables at their extreme values.

- `Full Factorial` - This method simulates all possible combinations.
- `Automatic` - This method calculates the worst case corners automatically without manually specifying methods. For more details, see Creating Worst Case Corners Automatically on page 109.

The first three algorithm methods, `OFAT 3-level`, `OFAT Sweep`, and `Central Composite Design`, are termed as Response Surface Methods (RSM), which means in these methods, a polynomial model is created and values for each corner variable is chosen based on the fact whether the specified value fits into the model. The worst case corner is the combination of variable values that are not simulated. As a result, after the worst case corner is created, the tool performs the corner validation for these methods.

For the `2^K Factorial` and `Full Factorial` methods, the tool determines the worst case corner from the simulated combination values. The combination that gives the worst result for each spec is identified as the worst case corner.

The following table outlines the comparison between these five methods that helps you choose an appropriate method to get the desired results:

Methods	**Accuracy**	**Simulation # (M parameters with N1, N2,...,Nm levels)**	**Example Simulation # (10 parameters with 3 level each)**	**Limitations**
OFAT 3-level	Quadratic and non-interacting variables	`2*M+1`	`21`	Not accurate for strong non-linear or interacting variables
OFAT Sweep	Non-linear variables	`N1+N2+...+NM-M+1`	`21`	Not accurate for interacting variables
Central Composite Design	Quadratic and variable interactions	`2*M+2*C(M,2)+1`	`111`	Not accurate for strong nonlinear variables
2^K Factorial	Linear and interacting variables	`2^M`	`1024`	Not accurate for non-linear variables

Full Factorial	Most accurate	`N1*N2*...*NM`	`59049`	Long simulation time for large number of variables

Example

To better understand these method, consider the following example:

Suppose you have three variables, `var1`, `var2`, and `var3`, with the following values:

`Var1: Lo, Nom, Hi`

`Var2: Lo, Nom, Hi`

`Var3: Val1, Val2, No, Val3, Val4`

OFAT 3-Level

If you run the `OFAT 3-Level` method, the table below describes the combination of variables that are simulated for this run.

Point	Var1	Var2	Var3
1	`Nom`	`Nom`	`Nom`
2	`Nom`	`Nom`	`Val1`
3	`Nom`	`Nom`	`Val4`
4	`No`	`Lo`	`Nom`
5	`No`	`Hi`	`Nom`
6	`Lo`	`Nom`	`Nom`
7	`Hi`	`Nom`	`Nom`

OFAT Sweep

If you run the `OFAT Sweep` method, the table below describes the combination of variables that are simulated for this run.

Point	Var1	Var2	Var3
1	`Nom`	`Nom`	`Nom`
2	`Nom`	`Nom`	`Val1`
3	`Nom`	`Nom`	`Val2`
4	`Nom`	`Nom`	`Val3`
5	`Nom`	`Nom`	`Val4`
6	`Nom`	`Lo`	`Nom`
7	`Nom`	`Hi`	`Nom`
8	`Lo`	`Nom`	`Nom`
9	`Hi`	`Nom`	`Nom`

Central Composite Method

If you run the `Central Composite Design` method, the table below describes the combination of variables that are simulated for this run.

Point	Var1	Var2	Var3
1	`Nom`	`Nom`	`Nom`
2	`Hi`	`Nom`	`Nom`
3	`Lo`	`Nom`	`Nom`
4	`Nom`	`Hi`	`Nom`
5	`Nom`	`Lo`	`Nom`
6	`Nom`	`Nom`	`Val4`
7	`Nom`	`Nom`	`Val1`
8	`Hi`	`Hi`	`Nom`
9	`Lo`	`Lo`	`Nom`

10	`Hi`	`Nom`	`Val4`
11	`Lo`	`Nom`	`Val1`
12	`Nom`	`Hi`	`Val4`
13	`Nom`	`Lo`	`Val1`

2^K Factorial

If you choose the `2^K FActorial` method, the table below describes the combination of variables that are simulated for this run

Point	Var1	Var2	Var3
1	`Lo`	`Lo`	`Val1`
2	`Lo`	`Lo`	`Val4`
3	`Lo`	`Hi`	`Val1`
4	`Lo`	`Hi`	`Val4`
5	`Hi`	`Lo`	`Val1`
6	`Hi`	`Lo`	`Val4`
7	`Hi`	`Hi`	`Val1`
8	`Hi`	`Hi`	`Val4`

Full Factorial

The `Full Factorial` method includes the 45 combinations of variables. This is the most accurate method.

Determining the Number of Data Points

The *No of Points* label to the right of the *Method* field displays the total number of data points for which simulation is to be run. The tool calculates these data points based on the algorithm selected in the Method drop-down list, nominal value and the range of parameter values. Nom, the tool dynamically changes the value of this label.

Importing Values from ADE XL Corners Setup

In the tabular format given in the centre of the form, specify the variables that you want to vary during the analysis. If the corners are already defined in the Corners Setup form in ADE XL, you can use the *Import from Corner Setup* button to directly import all or selected variables from the existing setup. When you click this button, the following two options appear:

- *Enabled Corners*—Select this option to import only enabled corners from the Corner Setup form.
- *All Corners*—Select this option to import all the corners that are present in the Corner Setup form.

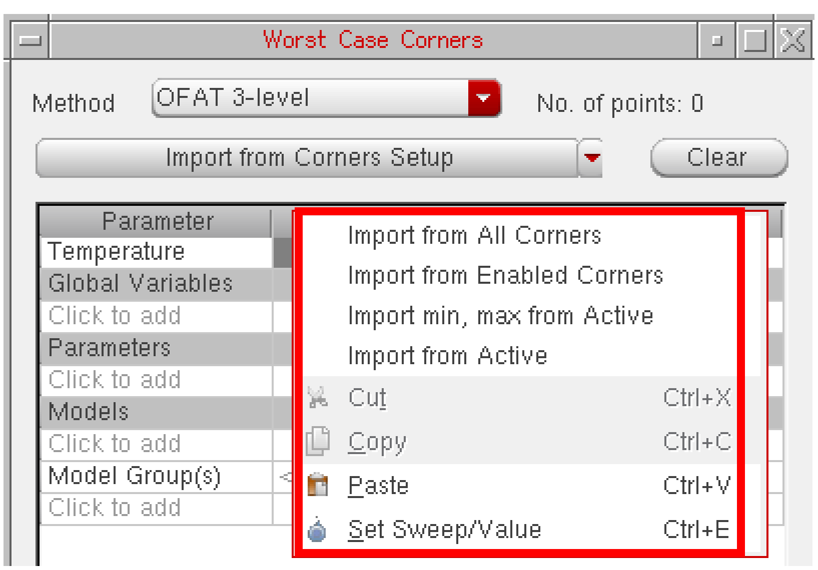
Note that this button follows the local corner selection, which means if corners are globally disabled in Run Summary, but enabled in the Corner Setup form, these corners are also imported in the Worst Case Corner form.

Tip

At any point, you can click the *Clear* button to clear all the content of the Worst Case Corner form.

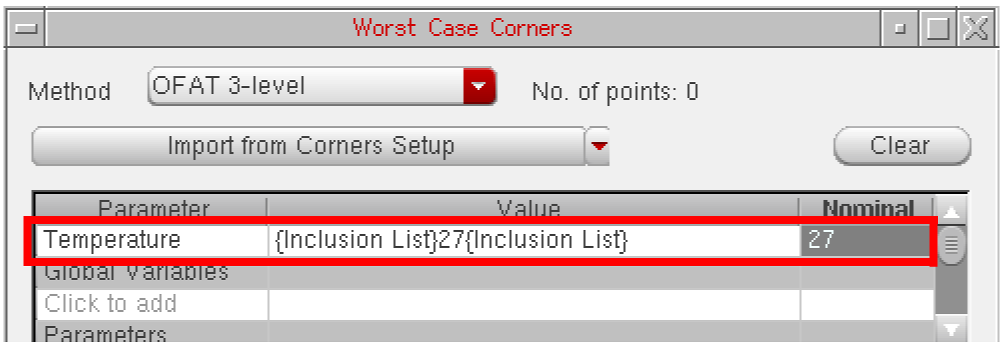
Specifying Temperature

To vary temperature, specify values for temperature in the Value cell. To import the value of temperature from all the corners, right-click in the *Value* column and choose *Import from All Corners* (refer the figure below).



Type a nominal temperature value in the *Nominal* cell for temperature or double-click in the cell to open the drop-down list. All the values specified in the *Value* cell are displayed in the list. Select a nominal temperature value from this list. Note that the nominal value should be in the given value list. If you type a nominal temperature value that is not in the specified value list, that value is not accepted by the form.

Alternatively, you can get the value of temperature from the test. For this, right-click in the *Nominal* cell for temperature and choose *Get Value from Test*. Note that if the value taken from the test does not exist in the *Value* cell, the tool automatically creates an inclusion list to add the nominal value to the existing values.



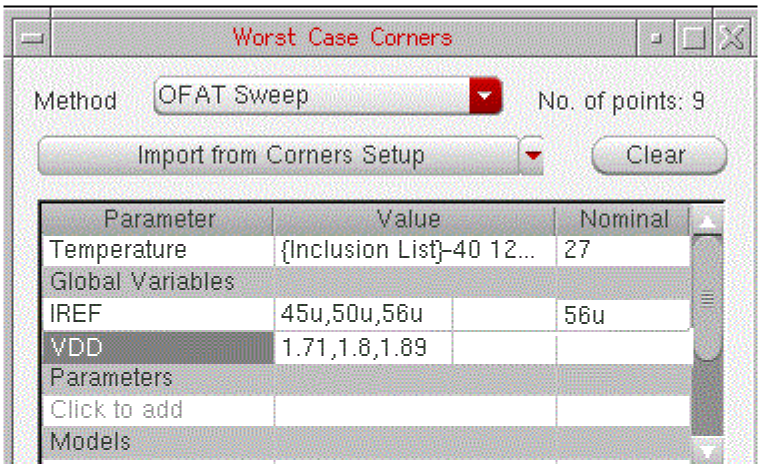
Specifying Global Variables

To specify global variable values:

➡ In the *Global Variables* field, click `Click to add`.

A drop-down list containing variables is displayed. Select the required variables from this list. The tool reads the range of values from the ADE XL setup and displays them in the *Value* column. The tool also reads the value of that variable from the test and if that value is one of the values in the *Value* column, displays the same in the *Nominal* cell.

Note: If you want to add all variables in the setup in a single step, select the `ALL` option from the drop-down list that appears when clicking `Click to add`.



For example, in the above figure, the value for *`IREF`* in the test is `56u`, which is there in the value list. Therefore, the tool automatically fills that value in the *Nominal* cell. However, for *`VDD`*, the value in the test is `2.0`, which is not in the given value list. Therefore, no value is displayed in the *Nominal* column. In this case, you can choose to get the nominal value either from the test, setup state, or from the reference point. For that, right-click in the *Nominal* column and choose an appropriate option. If the value taken from any of these sources does not exist in the *Value* cell, the tool automatically creates an inclusion list to add the nominal value to the existing set of values in the *Value* cell.

Note: If required, you can edit the value list in the *Value* cell. The tool dynamically calculates the number of points and changes the value in the *No. of points* label.

Specifying Parameters

To specify parameters:

➡ In the *Parameters* section, click `click to add`.

A drop-down list containing the parameters is displayed. Select the required parameters from this list. Similar to the global variables, the tool reads the range of values and design value for the selected parameters from the Variables and Parameters assistant and displays them in the *Value* and the *Nominal* column. Similar to the global variables, if the nominal value is not one of the values in the *Value* column, the *Nominal* cell is blank. You need to fill in that value.

Note: If there is only a single value in the *Value* column, the same value is filled in the Nominal cell as well. However, if required, you can change the nominal value. In that case, the tool creates an inclusion list with the values from both the *Value* and *Nominal* columns.

To change the parameter values, double-click in the *Value* column and edit the values by specifying a range or a list of space-separated values. Alternatively, you can select one or more values and right-click to display the shortcut menu and choose:

- *Import from All Corners* to import values defined for all corners in the Corner Setup form.
- Import from Enabled Corners to import values from defined for the enabled corners in the Corner Setup form.
- *Import Min, Max from Active* to load only the minimum and maximum values defined in the Data View or the Variables and Parameters assistant.
- Import from Active to load the values defined in the Data View or the Variables and Parameters assistant.
- Set *Sweep/Value* to edit the exclusion/inclusion list.

If required, you can change the nominal value for variables. The default nominal value is taken from the values of the parameter in the design. To change the nominal value for a variable or parameter, double-click in the *Nominal* cell for the variable or parameter. A list of possible nominal values is displayed. You can either select a value from the list or edit the value.

Note: If you specify a nominal value outside the specified range or the sweep set, the tool does not accept the value and makes the cell blank. A valid nominal value for a parameter is any value from within the range or sweep set corresponding to the parameter.

Specifying Model and Model Groups

To specify model files:

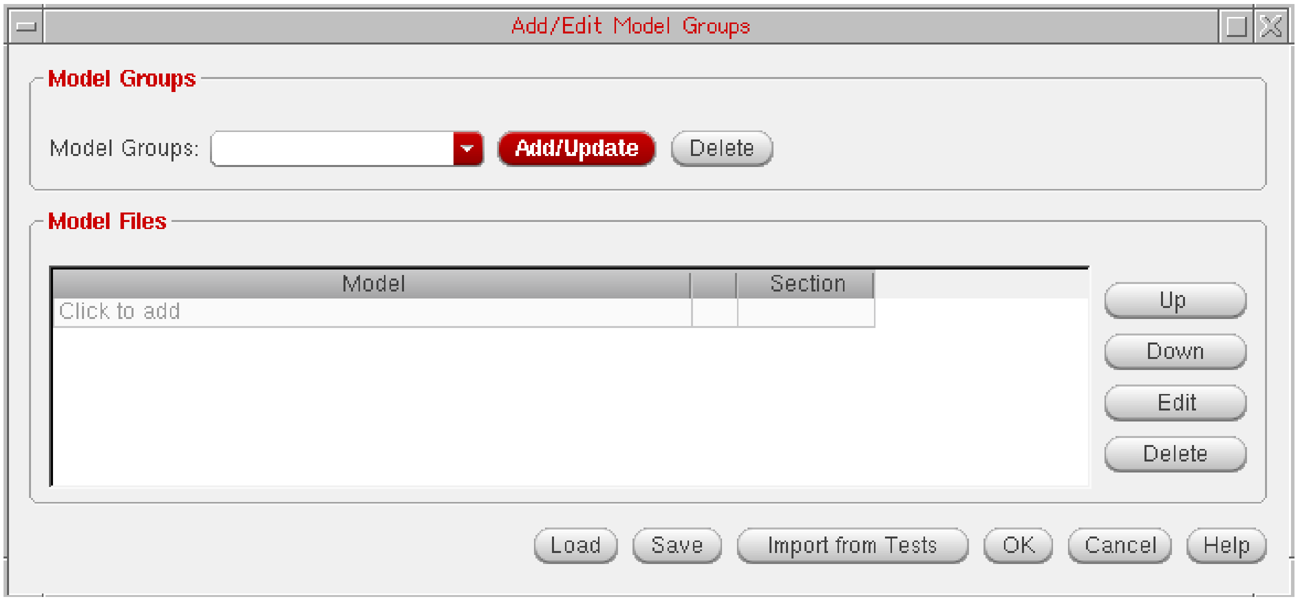
➡ Click `Click to add` in the Model section in the table.

The *Add/Edit Model* Files form appears, in which you can specify the model files.

To specify model groups:

➡ Click `Click to add` in the Model Group(s) section in the table.

The *Add/Edit Model Group* form appears, as shown in the figure below:



- ❑ In the *Model Groups* drop-down, select the model groups that you want to vary for the simulation.
- ❑ In the *Model Files* panel, browse and select a model file for which you want to vary different sections. The name of the file appears in the cell.
- ❑ Select a section that you want to add to the list. You can select multiple sections that you want to vary for the model file.
- ❑ Click the *Import from Tests* button to import the model files directly from the test.
- ❑ Click *OK*.

For more information about the fields in this form, see Adding Model Files to a Corner in the *Virtuoso Analog Design Environment ADE XL User Guide*.

To add a nominal value, double-click in the *Nominal* cell and select name of a section that you want to use as a nominal value for the selected model file.

Note: It is also possible to specify a model file without selecting any section. In this case, the model file specified in the Worst Case Corners form is used instead of the model file specified with the test.

The setup required for creating worst case corners is complete.

Running Worst Case Corners

Important

The worst case corners cannot be run with swept variables. Therefore, either disable sweeps or ensure that all the swept variables are overridden in the Worst Case Corners form.

After the worst case corner variables and methods have been specified in the Worst Case Corner form, click the *Run Simulation* button on the *Run* toolbar to run the simulation for the specified corners.

Below is a step-wise description about the basic tasks that are performed when a worst case corner simulation is run and the worst case corners are evaluated. These steps help you find out whether the design meets the specification across all corners before running a large number of simulations.

1. In the first step, the worst case corners are created by generating and simulating the design samples for the specified set of parameter values. You can use different algorithms to generate the worst case corners. For more information about the available algorithms, see Choosing a Method on page 95.
2. Then, the simulation is run for the specified parameter and values. If you have chosen an RSM algorithm in the Worst Case Corner form, the tool generates the predicted results and validates the obtained simulation results against the predicted results. For more information about the results validation, see Validating Worst Case Corners on page 105.
3. Now, the simulated results generated from the previous step are compared with the predicted result expressions. If the generated results do not match the predicted results, you can use a different algorithm and rerun the worst case corner simulation to achieve the desired results. This makes the process faster because you are not required to run all the simulations again if you use an RSM algorithm.
4. The above steps are then repeated to predict the values for all remaining corners. All the expressions are evaluated and then the simulation is run for all the combination sets.

Validating Worst Case Corners

If you have selected an RSM algorithm, such as OFAT 3-Level, OFAT Sweep, or Central Composite Design, ADE GXL generates a set of predicted results and then validates these results against the results obtained from the actual simulation run.

Refer to the following sections to know more about how the worst case corners are validated:

- Viewing Validation Results Using Run History on page 106

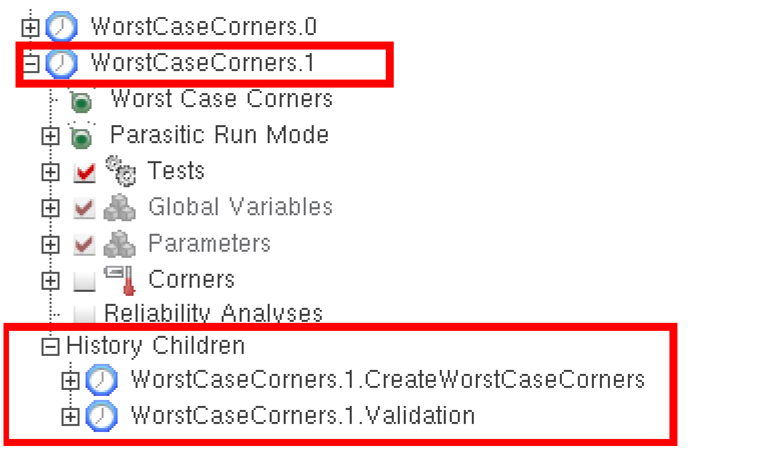
- Viewing Validation Results Using Log File on page 108

Viewing Validation Results Using Run History

To view the history information for a particular simulation run, click the *History* tab. The histories of the previous simulation run are displayed in the Data View assistant. The history name for a particular simulation run includes the name of the run mode.

Alternatively, you can click the history tabs displayed at the bottom of the Results window.

To view the history for a previous worst case corner run, expand *WorstCaseCorners.1* as shown in the figure below.



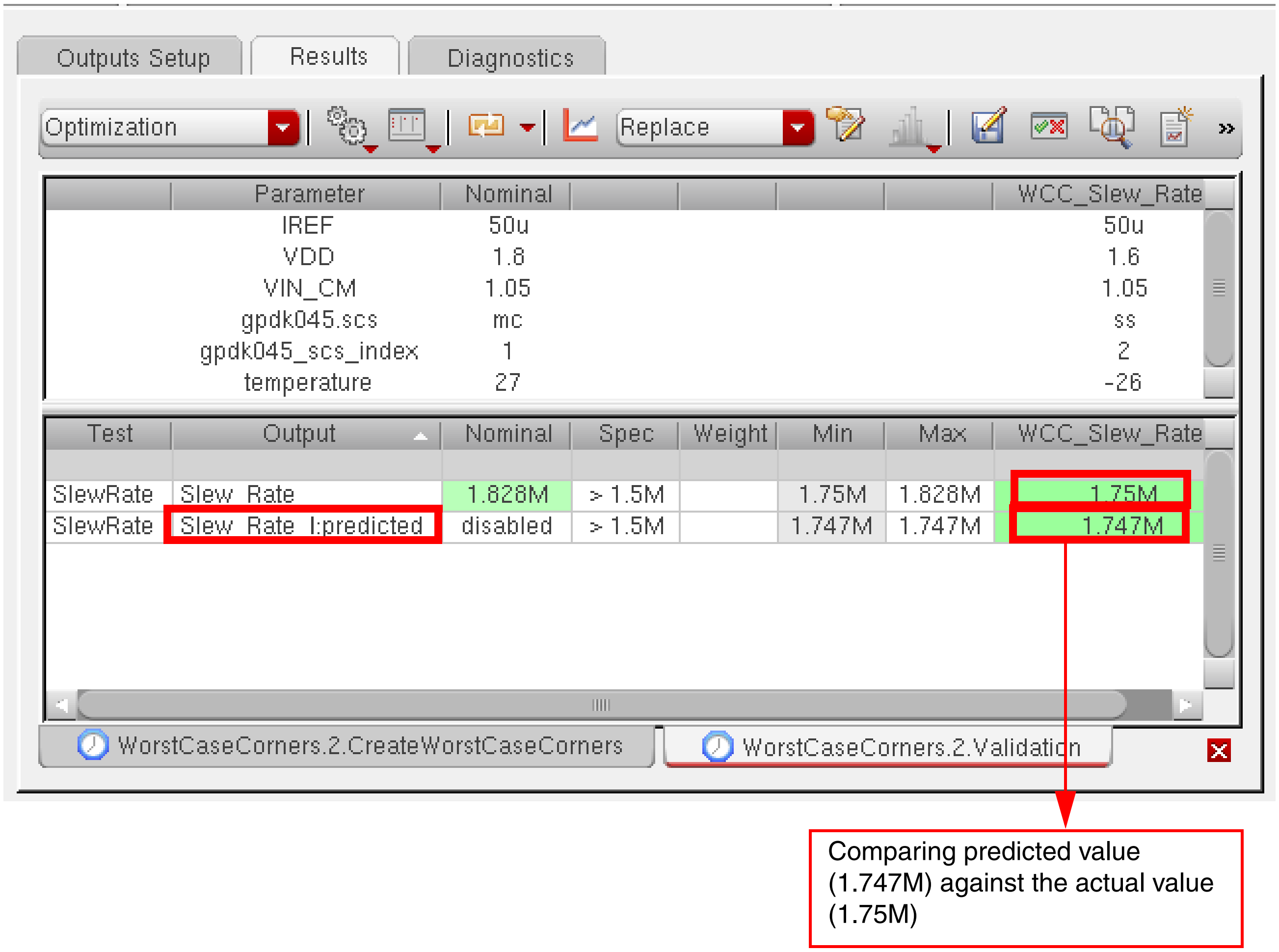
If you selected an RSM method, such as OFAT 3-level, OFAT Sweep, or Central Composite, the worst case corner simulation run creates a group run with child histories. To view child histories, expand the *History Children* section as shown in the figure above. The following two histories are displayed under this section:

- *WorstCaseCorners.1.CreateWorstCaseCorners*
- *WorstCaseCorners.1.Validation*

In the first child run, the samples are generated and simulated to create the worst case corner. This run also generates the predicted result expressions for each corner if an RSM algorithm is used.

In the second child run, the worst case corners are simulated and validated against the predicted results. The validation is performed only if you have selected a RSM algorithm in the Worst Case Corners form.

In the second child history tab, the predicted worst case corner value is calculated for all corners and is displayed for each test in a separate row in the results window. The figure below shows the predicted and generated corners for `SlewRate`.

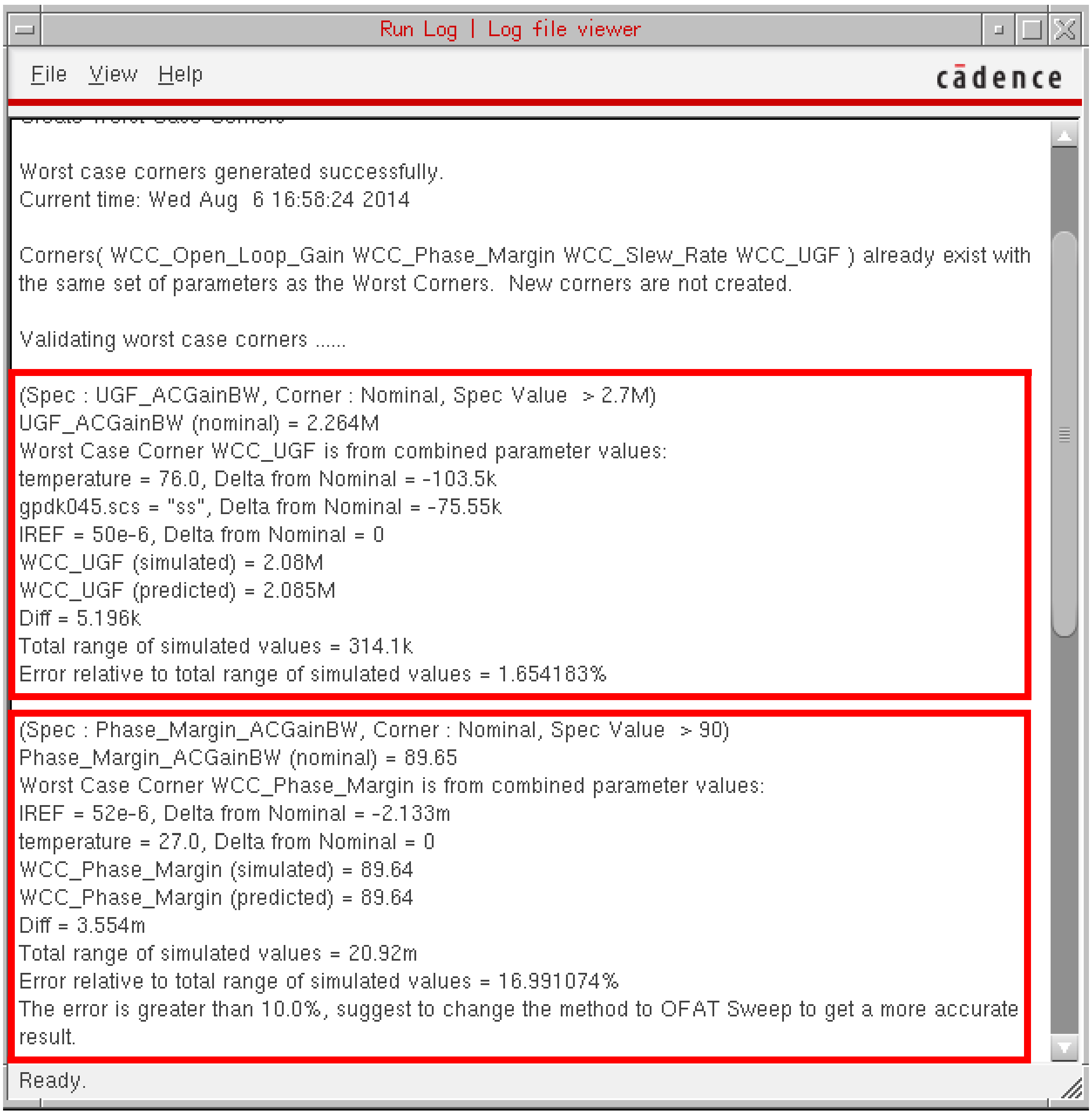


In this figure, the *Output* column displays two values for the Test, *SlewRate*, `Slew_Rate` and `Slew_Rate_I:predicted`.

In the next step, the simulated value for the test that has been generated after simulating the worst case corners is compared against the corresponding predicted value, and then the difference between these two values is calculated. If the test output includes a range spec, two worst case corners are created, one for the high value range and the other for the low value range. Similarly, two predicted results values are generated for high and low value ranges, respectively.

Viewing Validation Results Using Log File

A log file is created while a simulation for the worst case corners is running. This log file maintains a record of all the tasks that the tool performs during the worst case corner run (as shown in the figure below). This log file displays the difference between the actual simulated value and the predicted value and the error relative to the total range of simulated values, which provides you a hint about whether the worst case corner run was successful.



If the relative error is greater than `10%` of the total range of simulated values, the tool displays a message in the log file that suggests you to use another alogrithm to get more accurate worst case corners. For example, in the figure above, the tool suggests you to use the OFAT Sweep algorithm to get the optimal results. You can then change the method in the Worst Case Corners form and rerun the simulation.

The approach described above is followed for all set of corners.

Creating Worst Case Corners Automatically

The `Automatic` method enables you to find worst case corners without manually specifying the algorithm to generate accurate worst case corners. It accurately and efficiently identifies worst cases corners for a wide range of applications. If you do not know which algorithm you need to apply to find worst case corners, it is recommended that you use the Automatic method, which finds all the worst case corners efficiently and generates worst case corners accurately by running fewer simulations.

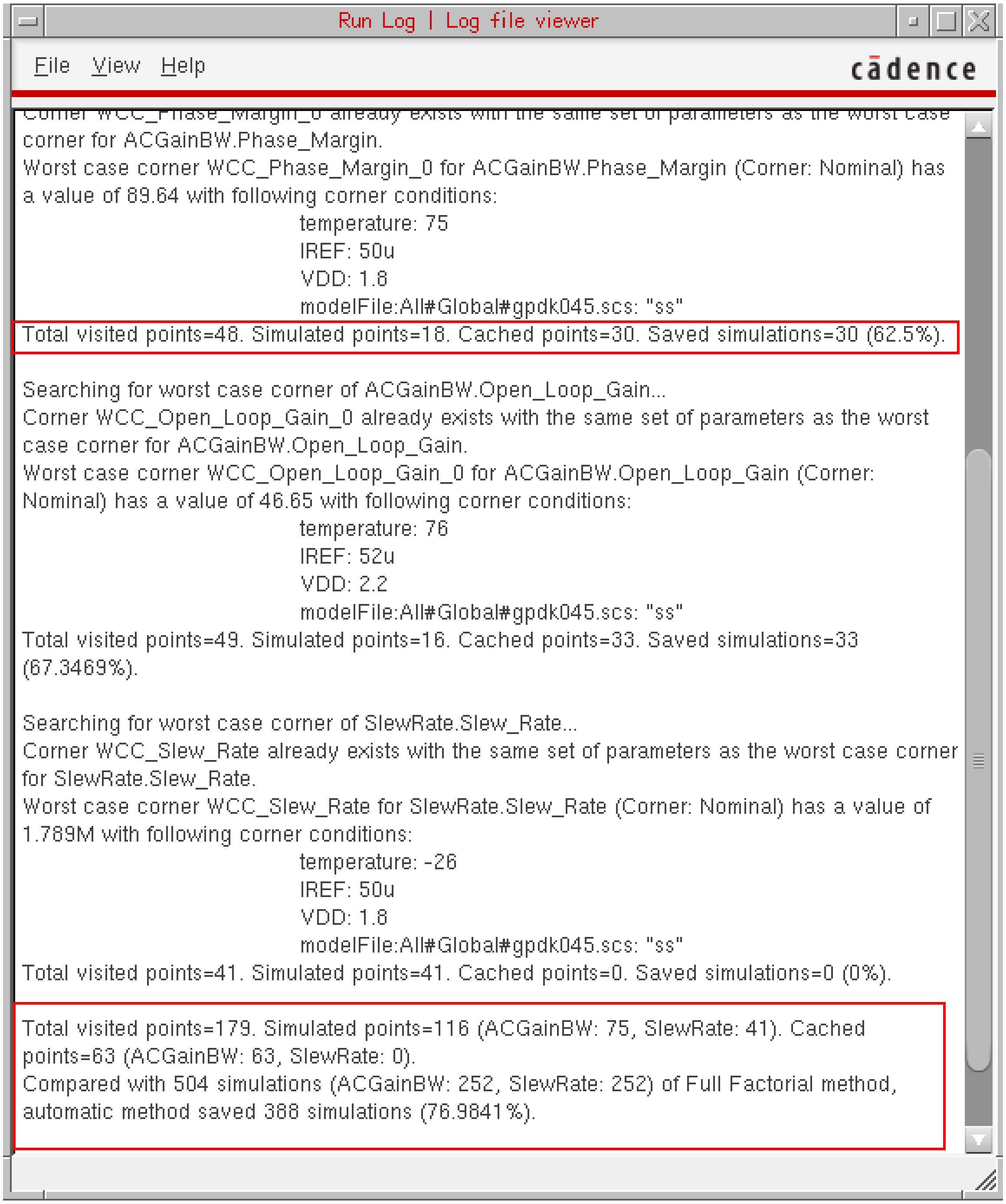
To generate worst case corners using this method, select the `Automatic` option from the *Method* drop-down list in the *Worst Case Corner* form.

When you use this method, the ADE GXL performs the following tasks:

1. Runs OFAT sweep method.
2. Runs fractional factorial design on the important variables based on OFAT results.
3. Runs local optimization based on the worst case corner from OFAT and fractional factorial samples.

The worst case corners are generated and displayed in the Corners Setup form. For more details about different ways of viewing worst case corners, see Viewing Worst Case Corners on page 111.

The figure below displays the run log generated after the simulation run is complete.



Notice the additional information displayed in the run log about the number of simulations performed during the Worst Case Corners run for each specification. For example, when searching for the worst case corner for `Phase_Margin`, 48 points were visited by the algorithm. Out of these, only 18 points required new simulations to run. The simulations for this ADE XL test were cached when searching for the worst case corner for another specification, such as UGF.

From the information displayed at the end of the run log, you can determine that the `Automatic` method required a much lesser number of simulation runs (116) as compared to the `Full Factorial` method (504).

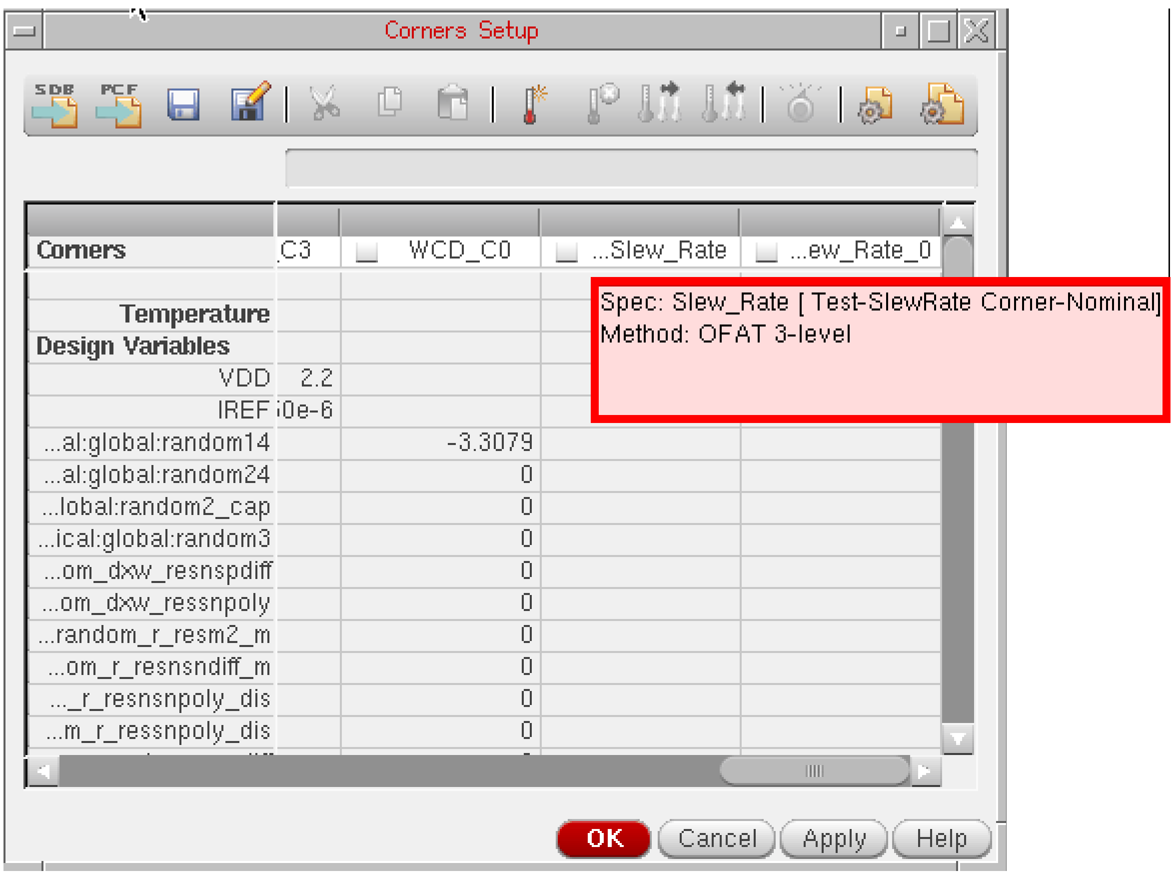
Viewing Worst Case Corners

Worst case corners are created for every specification and are indicated by the prefix `WCC`. You can view worst case corners by doing one of the following:

- Viewing Worst Case Corners in the Corners Setup Form
- Viewing Worst Case Corners in the Data View
- Viewing Results in Sensitivity Analysis Window on page 113

Viewing Worst Case Corners in the Corners Setup Form

In the Corners Setup form, a tooltip displays the spec and method that has been used to create the worst case corners. Move the cursor on a worst case corner name to view the tooltip, as shown in the figure below.

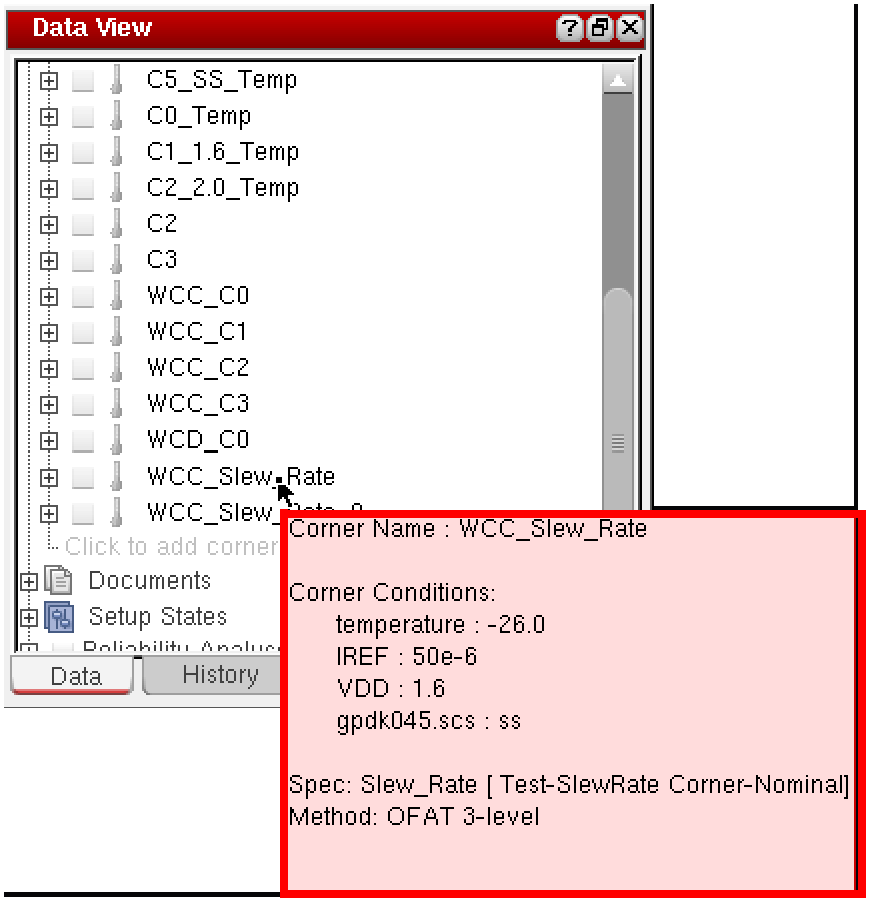


Viewing Worst Case Corners in the Data View

To view the worst case corners from the Data View assistant pane, follow these steps:

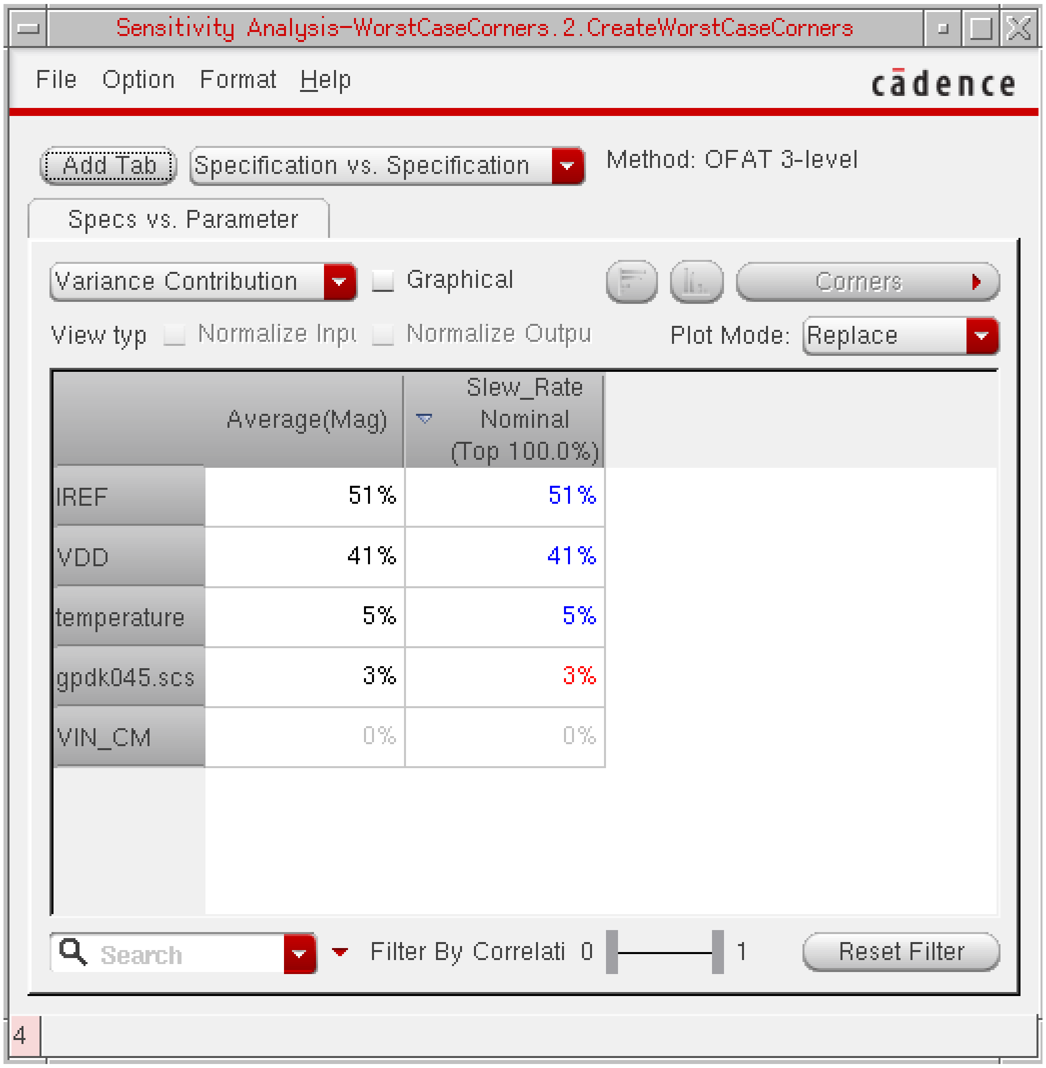
1. In the Data View assistant pane, click the *Data* tab.
2. Expand the *Corners* tree.
3. Point to a worst case corner name, such as `WCC_SlewRate`.

A tooltip, as shown in the figure below, displays the spec and method used for creating the worst corners.

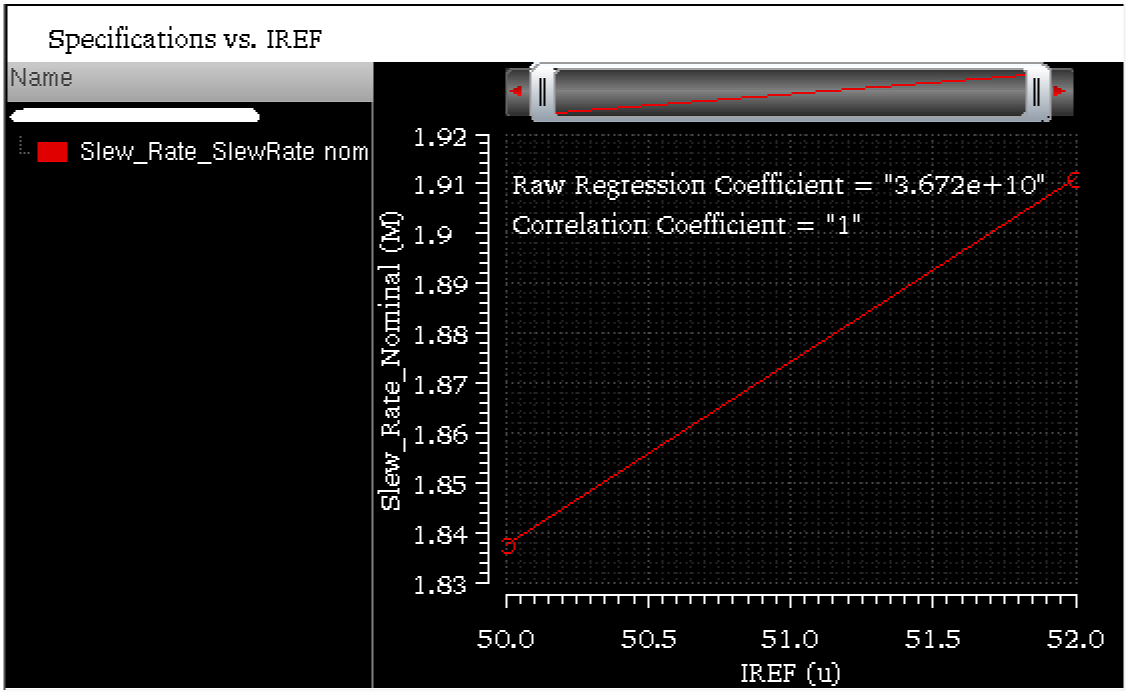


Viewing Results in Sensitivity Analysis Window

The Sensitivity Analysis window opens up automatically when the simulation run is complete. This window displays the results generated from the actual simulation data. The results, which are the relationship between each measurement and the parameter that you change, are displayed in a separate row. Note that by default the result is displayed in the percentage format.



When you double-click a value displayed in the cell, the Virtuoso Visualization and Analysis XL window appears displaying the plots for the selected measurement value. For example, if you double-click the *IREF* and *Slew_Rate Nominal* cell, the following graph shown in the below figure appears. Note that this graph displays the regression and correlation coefficient values for the selected measurement results.



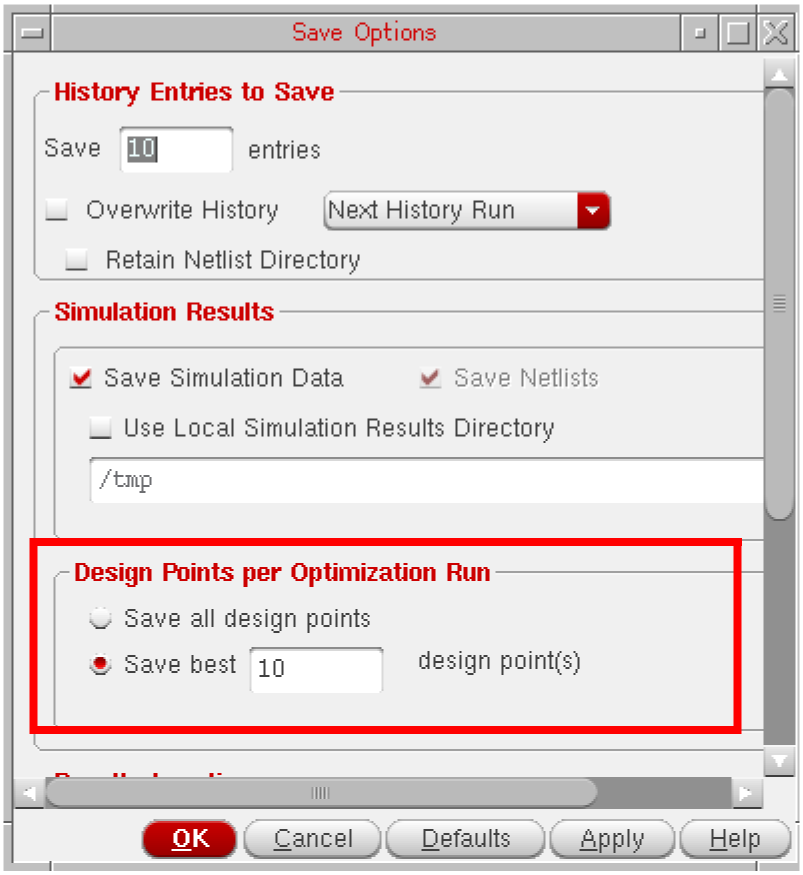
Specifying the Number of Best Design Points to be Saved

When you specify the number of design points that are to be saved during a optimization run, the same setting is applied to the worst case corners run. This means only the specified number of best design points are saved in the database after the run is complete and not all the design points.

To specify the number of best design points that are to be saved during a optimization or worst case corners run:

➡ Choose *Options – Save*. The *Save Options* form appears.

In this form, in the *Design Points per Optimization Run* section, select the *Save best design point(s)* check box and specify the number of best design points to be saved.



Alternatively, to save all the design points in the database, select the *Save all design points* option.

4

Circuit Optimization

In addition to parameterizing a design for sizing and optimization using ADE variables, you can also use a parameterization flow in ADE GXL. To use this flow, you can parameterize a design and set specifications, then perform a local or global optimization and back-annotate the results of the optimization.

This chapter covers the following tasks:

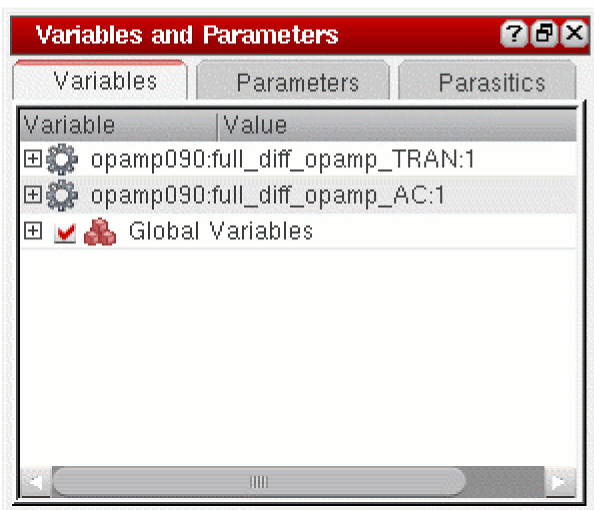
Parameterizing the Design

When you parameterize the design for optimization, you capture the critical device relationships in the circuit. For example, the input transistors of an opamp must be matched, and the transistors of a current mirror must be ratioed. In order to run optimization, you must define a range of legal values for optimization for each transistor that is the "master" in a matching relationship.

To begin parameterizing the design.

➤ Choose *Window — Assistants — Variables and Parameters*.

The Variables and Parameters pane appears.



Matching Devices and Device Properties

You have three options for matching. You can match all device properties for a device, or you can match only specific properties. You can also ratio-match. When you're finished specifying device relationships, you can define legal values for device properties.

Matching Devices

To match devices:

1. Select the devices you want to match in the schematic.

   The devices are listed on the upper half of the Parameters tab of the Variables and Parameters assistant pane.

2. In the Parameters tab, select the device you want as the “master device.” The master device is the device that all the other selected devices will match.
3. Click the *Match Parameters* button.

   The matched parameters appear on the lower half of the Parameters tab of the Variables and Parameters assistant pane.

You can now modify the values of matched parameters to vary them during the optimization process.

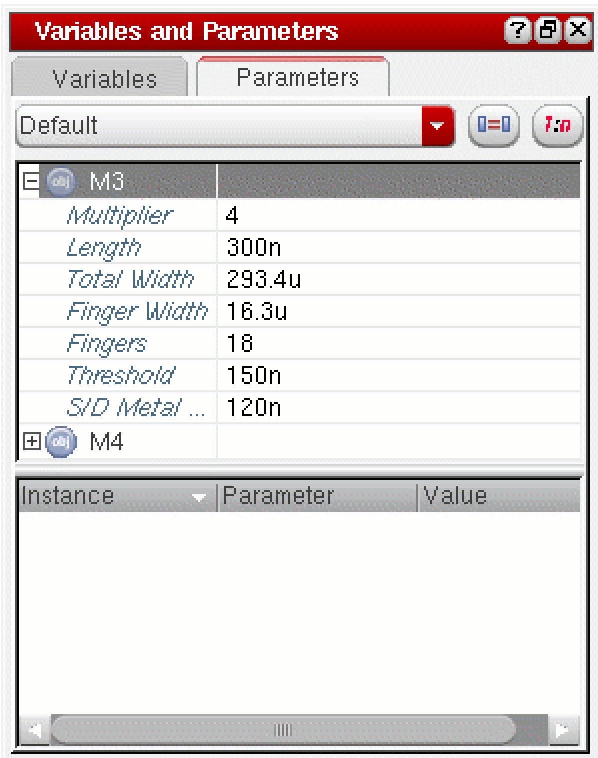
Matching Device Properties

To match specific device properties:

1. In the schematic, click the devices for which you want to match properties.

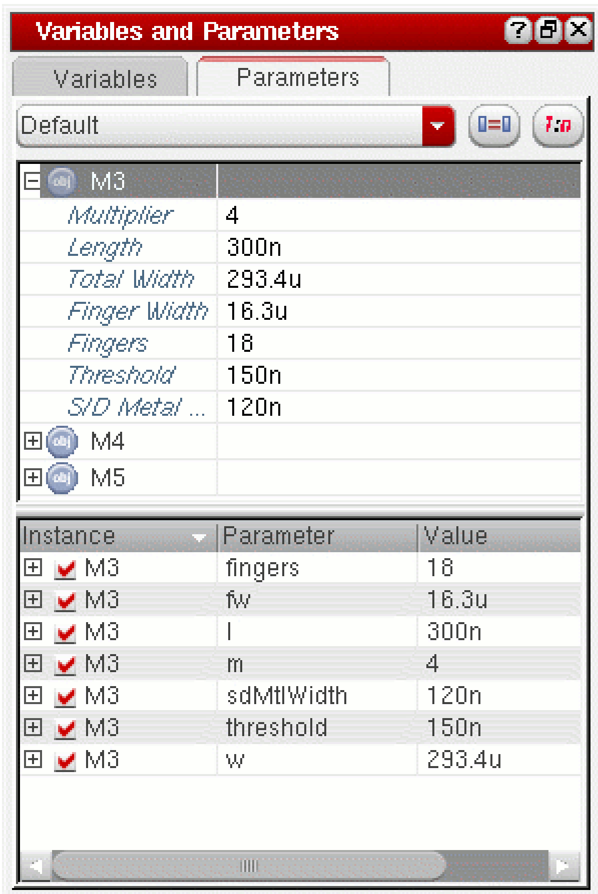
   As you click, each device and its parameters appear on the upper half of the Parameters tab of the Variables and Parameters assistant pane.
2. Ensure that *Default* is selected in the drop-down list on the upper half of the Parameters tab of the Variables and Parameters assistant pane.
3. Click the + sign next to a device with whose properties you want to match the properties of other devices.

The list of properties for the device appears, as well as the schematic value of each property.



4. Select the property you want to match.
5. Click the *Match Parameters*  button.

The matched parameters appear on the lower half of the Parameters tab.



You can now modify the values of matched parameters to vary them during the optimization process.

Ratio-Matching Device Properties

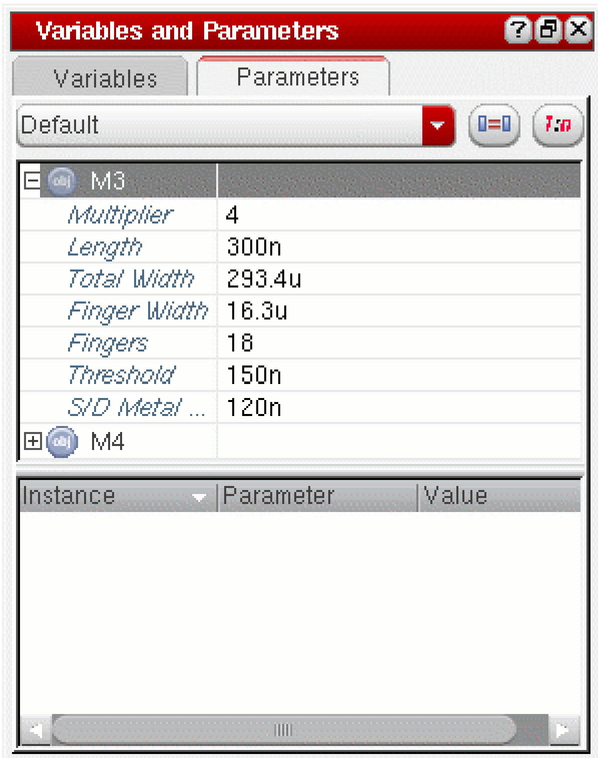
To ratio-match specific device properties:

1. In the schematic, click the devices for which you want to ratio-match properties.

   As you click, each device and its parameters appear on the upper half of the Parameters tab of the Variables and Parameters assistant pane.

2. Ensure that *Default* is selected in the drop-down list on the upper half of the Parameters tab of the Variables and Parameters assistant pane.

3. Click the + sign next to a device with whose parameters you want to ratio-match the parameters of other devices.



4. Select the parameters you want to ratio-match.
5. Click the *Ratio Matched Parameters*  button.

   ADE GXL automatically ratios the parameters based on the lowest value.

   The ratio-matched parameters appear on the lower half of the Parameters tab.

You can now modify the values of ratio-matched parameters to vary them during the optimization process.

Defining Values for Optimization

To define a legal range of values for devices:

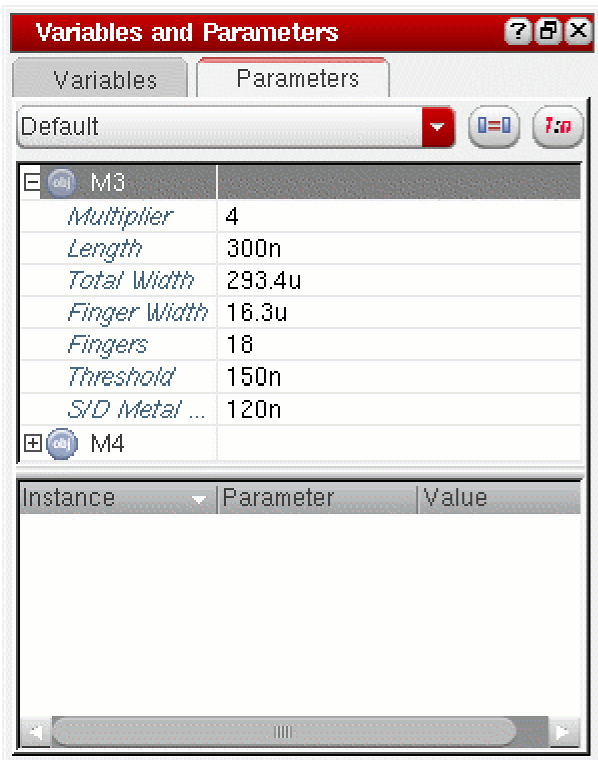
1. In the schematic, click the devices for which you want to specify values.

   As you click, each device and its parameters appear on the upper half of the Parameters tab of the Variables and Parameters assistant pane.

2. Ensure that *Default* is selected in the drop-down list on the upper half of the Parameters tab of the Variables and Parameters assistant pane.

3. Click the + sign next to a device whose parameters you want to modify for optimization.

   The list of parameters for the device appears, as well as the schematic value of each property.



4. Click on the value for the parameters you want to modify and type a range or list of values. Specify ranges in the format `MIN:STEP:MAX`. You can separate a list of values with either commas or spaces.

   The modified parameters appear on the lower half of the Parameters tab.

Setting Up Specifications

When you set specifications on outputs, you establish the performance specifications that ADE GXL must meet during synthesis. These specifications are used as benchmarks for candidate circuit quality during optimization.

You can set up general specifications, area constraint specifications and operating region specifications. For more information, see the following topics:

- General Specifications on page 124
- Device Area Constraint Specifications on page 127
- Running Optimization on page 132

General Specifications

For each specification, you specify an objective, or performance constraint, then specify a target value for that specification.

Objective Types

There are five types of objectives:

- open-ended
- close-ended
- range
- tolerance
- info

An open-ended specification is when you choose either minimize (min) or maximize (max), then specify an ideal value. For an open-ended specification, ADE GXL tries to continually improve upon the ideal value. So, if the ideal value for the *ugf* specification is maximize to 1e+8, 3e+8 is better than 2e+8.

A close-ended specification is when you specify a target value, then specify that the resulting specification must be greater than (>) or less than (<) that target value. For a close-ended specification, ADE GXL attempts to meet the target value, and exceeding the target is no better than meeting the target. So, if the target value for *ugf* is >1e+8, 3e+8 is no better than 2e+8.

You can also specify a range specification, in which you specify both an upper and a lower boundary for the target value. For a tolerance (tol) specification, you specify a target value and an allowable percentage deviation.

The optimizer treats the range and tolerance specs as closed-ended unless all the specifications are met. When all the specifications are met, range and tolerance are treated as open-ended. When all the specifications are met and the `all specs met` stopping criteria has not been specified, the optimizer continues to prefer the values closer to the midpoint of the range or tolerance target value.

You can specify an info specification when you want to see the value of a measurement, but don't want that specification to affect sizing.

Design specifications might include specifying an open loop gain maximized with an acceptable value of 60dB, or a settling time < 15ns.

Tip

You can also specify area goals. For more information, see "Device Area Constraint Specifications" on page 127.

Specification Weights

ADE GXL supports weighting on specifications, that is, the ability to put particular emphasis on one or more specifications. You may use weighting if you have certain important specifications, and it is imperative that ADE GXL meet those specifications, even at the expense of other specifications. Or, if you have run ADE GXL without any specification weights, and you have one particular specification that is not being met, you can put emphasis on that specification in order to ensure that ADE GXL meets it.

To set specification weighting, you specify a positive integer as the "weight." ADE GXL multiplies the cost function for that specification by the integer specified for the weight. Because this specification is now "x" times more important than other specifications, it changes the way ADE GXL searches the design space. ADE GXL will put particular stress on meeting that specification, even at the expense of specifications weighted at "1." This discrepancy is especially true of specifications that are minimized or maximized, because ADE GXL is already working harder to optimize the value for that specification.

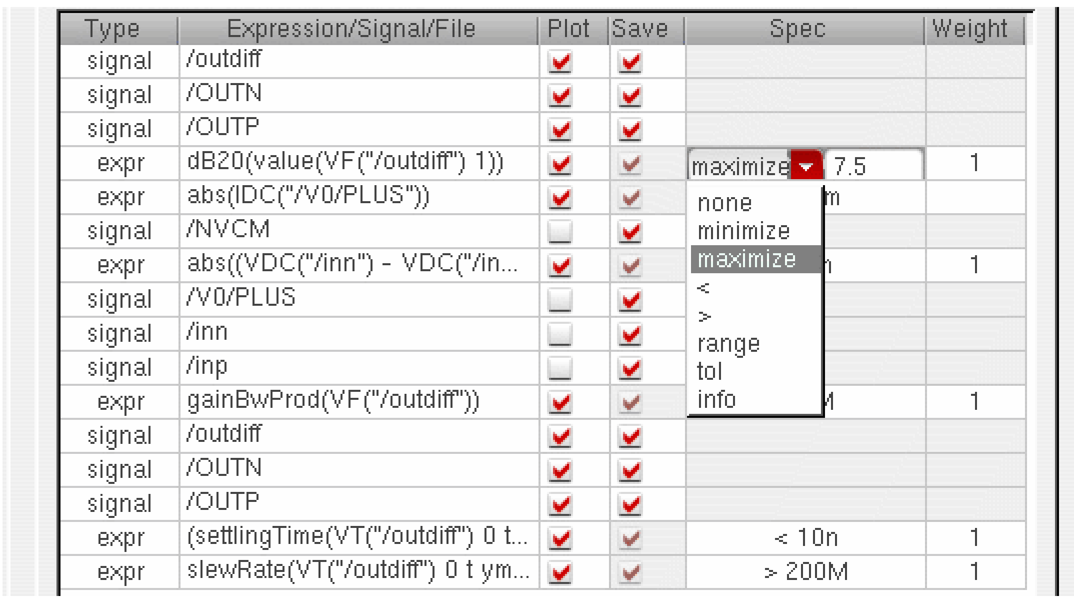
In general, ADE GXL is able to find a working solution without specification weighting. As a result, Cadence advises that you run ADE GXL first with all specifications weights set to "1." If ADE GXL is repeatedly unable to meet a particular specification, you can put more emphasis on meeting that specification using specification weighting.

Setting Up Specifications

To set up specifications:

1. In the Outputs Setup tab, double-click on the *Spec* column in the row for the output for which you want to set up specifications.

   A drop-down list of objectives appears.

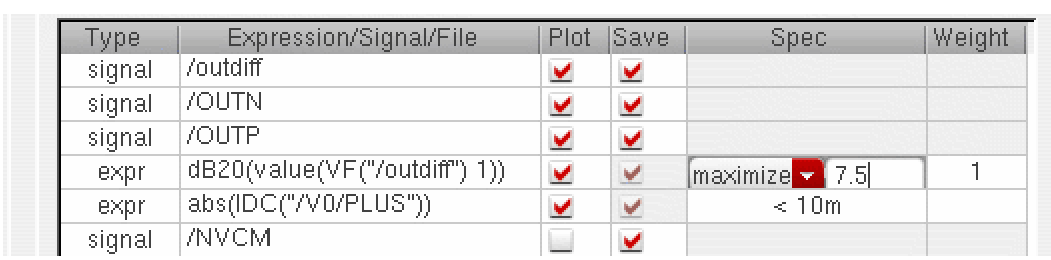


2. Choose an objective from the drop-down list.

   For more information on the types of objectives, see Objective Types.

3. Select the field next to the drop-down list.

4. Enter a target value in the field.



5. (Optional) Specify a weight for the specification in the Weight column.

   For more information on weights, see Specification Weights.

6. Repeat step 1 through step 5 for all outputs for which you want to set specifications.

Device Area Constraint Specifications

You can specify area constraint specifications for use in optimization runs. For each device, you can either manually specify an area formula, or use a CDF parameter to set default area formulas for devices. For more information about using a CDF parameter to set default area formulas, see Setting Default Area Formulas on page 127.

Tip

You can add multiple area constraint specifications for a test if you want to set different specifications for some devices.

For more information, see the following topics:

- Setting Default Area Formulas on page 127
- Setting Up Area Constraint Specifications on page 128
- Modifying an Area Constraint Specification on page 130
- Copying a Device Formula on page 130
- Including Devices in the Area Constraint Specification on page 130
- Excluding Devices from the Area Constraint Specification on page 131
- Deleting Devices with Area Formulas on page 131

Setting Default Area Formulas

You can specify default area formulas for devices using the `AreaFormula` CDF parameter.

If you want to use a different CDF parameter name for specifying default area formulas, specify the parameter name using the *`defAreaProp`* environment variable in one of the following ways:

- In the CIW, enter the following:

```
envSetVal("adexl.gui" "defAreaProp" 'string “<myParamName>”)
```

- In your `.cdsenv` file, enter the following:

```
adexl.gui defAreaProp string “<myParamName>”
```

For more information about CDF parameters, see the Component Description Format User Guide.

Setting Up Area Constraint Specifications

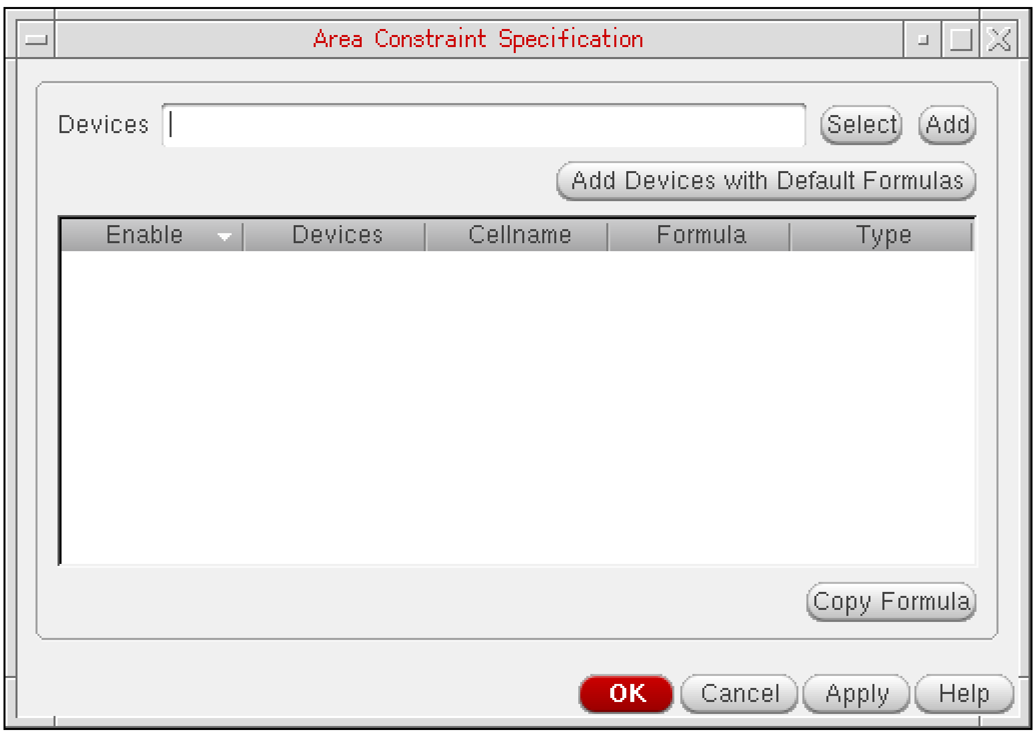
To set up area constraint specifications, do the following:

1. On Outputs Setup tab, click the *Add new output*  button.
2. In the drop-down list, select a test and choose *Area Specification*.

Tip

You can also right-click on a test name in the Outputs Setup tab and choose *Add Area Specification* to set up area constraint specifications for that test.

The Area Constraint Specification form appears.



3. Specify a device name for which you want to add an area formula:
   - ❑ Select a device in the schematic. To do this, click the *Select* button to display the schematic window. Select the device in the schematic window and press the *Esc* key. The device name is displayed in the *Devices* field.
   - ❑ Enter the device name in the *Devices* field.
4. Click *Add* to add the device to the list.
5. Select the *Formula* field for the device and enter an area formula.

You must enter an arithmetic combination of valid CDF parameters that are specified for the device.

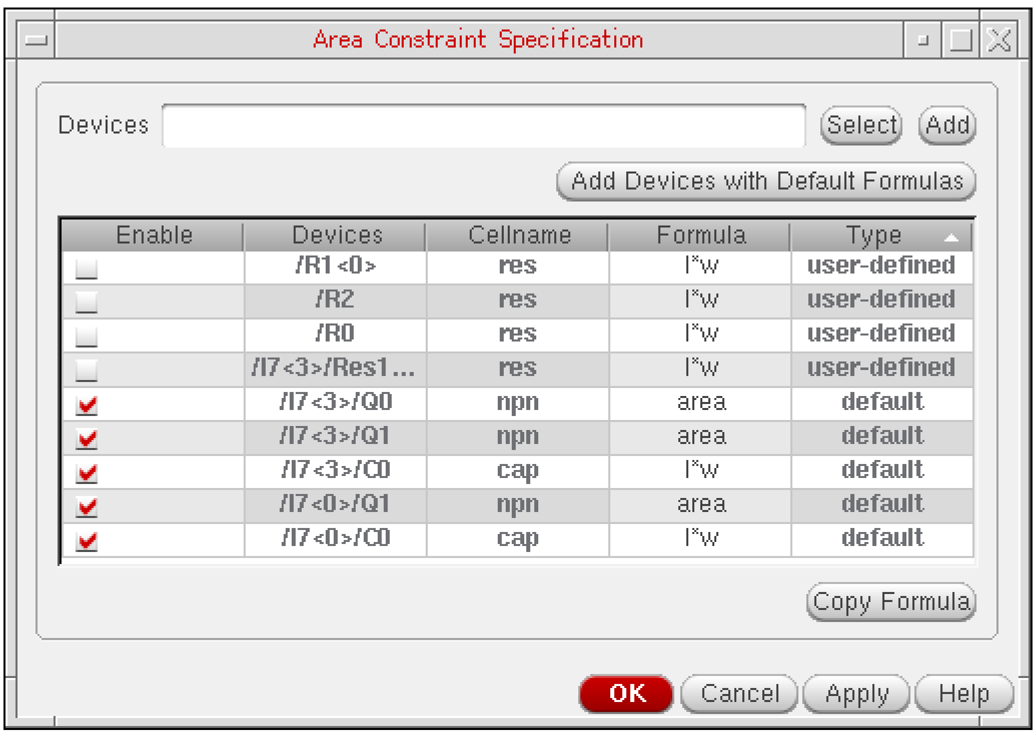
Tip

You can use the Copy Formula button to copy area formulas from one device to another.

6. Repeat step 3 through step 5 for any other devices you want to include in the area constraint specification.

7. Click the *Add Devices with Default Formulas* check box to add all instances in the design for which default formulas are specified using the `AreaFormula` CDF parameter. For more information, see Setting Default Area Formulas on page 127.

   Note: You can modify any default formula by selecting the *Formula* field and typing a new area formula.

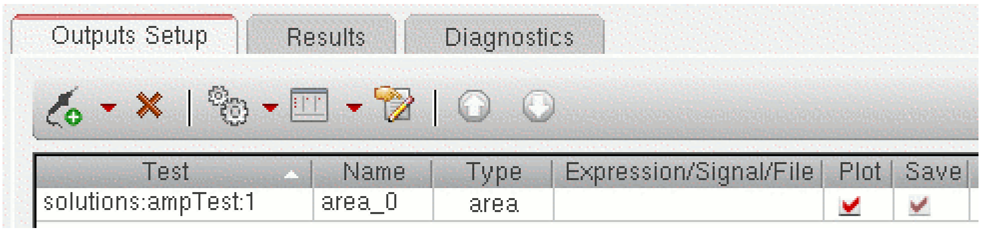


8. To include a device in the area constraint specification, select the *Enable* check box for the device. De-select the *Enable* check box for the devices you do not want to include in the area constraint specification.

   Note: All devices with default area formulas are included by default, while any devices for which you manually entered area formulas are not.

9. Click *OK*.

   The area constraint specification is displayed in the Outputs Setup tab.



   The area constraint specification is assigned the name `area_seqNum`, where `seqNum` is `0` (zero) for the first area constraint specification you add. You can double-click on the *Name* field and modify the name.

10. Set up specifications for the area constraint specification. For more information, see Setting Up Specifications on page 126.

Modifying an Area Constraint Specification

To modify an existing area constraint specification:

1. In the Outputs Setup tab, double-click on the *Expression/Signal/File* field for the area constraint specification.

   The Area Constraint Specification form appears.

2. Modify the area constraint specification as required.

Copying a Device Formula

To copy the area formula from one device to another:

1. Select the device from which you are copying the formula.
2. Control-click on the device to which you want to copy the formula.
3. Click *Copy Formula*.

Including Devices in the Area Constraint Specification

➡ To include a device in the area constraint specification, select the *Enable* check box.

To include more than one device at once:

1. Select the devices you want to include.
2. Right-click and choose *Enable*.

Tip

You can enable all devices by right-clicking and choosing *Enable All*.

Excluding Devices from the Area Constraint Specification

➡ To exclude a device from the area constraint specification, deselect the *Enable* check box.

To exclude more than one device at once:

1. Select the devices you want to exclude.
2. Right-click and choose *Disable*.

Tip

You can disable all devices by right-clicking and choosing *Disable All*.

Deleting Devices with Area Formulas

To delete devices and their area formulas:

1. Select one or more devices.
2. Right-click and choose *Delete*.

Running Optimization

You have three options for optimization:

Local optimization runs the schematic design, then deterministically improves that design. Local optimization is useful if:

- You have a starting design that you want to improve.
- A design meets specifications at the nominal corner but not across all corners.
- Models in a PDK have changed just enough to cause the design to no longer meet all specifications.

Note that local optimization only searches the design space around the specified point, so ADE GXL may not locate the best point possible to meet the design specifications. In addition, local optimization does not guarantee small perturbations to your variable values.

Unlike local optimization, global optimization does not require a reference point. It efficiently searches over all of the variables defined to find a good solution for your design.

Tip

If you have a large number of corners, you may want to run Size Over Corners instead of running global or local optimization. For more information, see Sizing Over Corners on page 146.

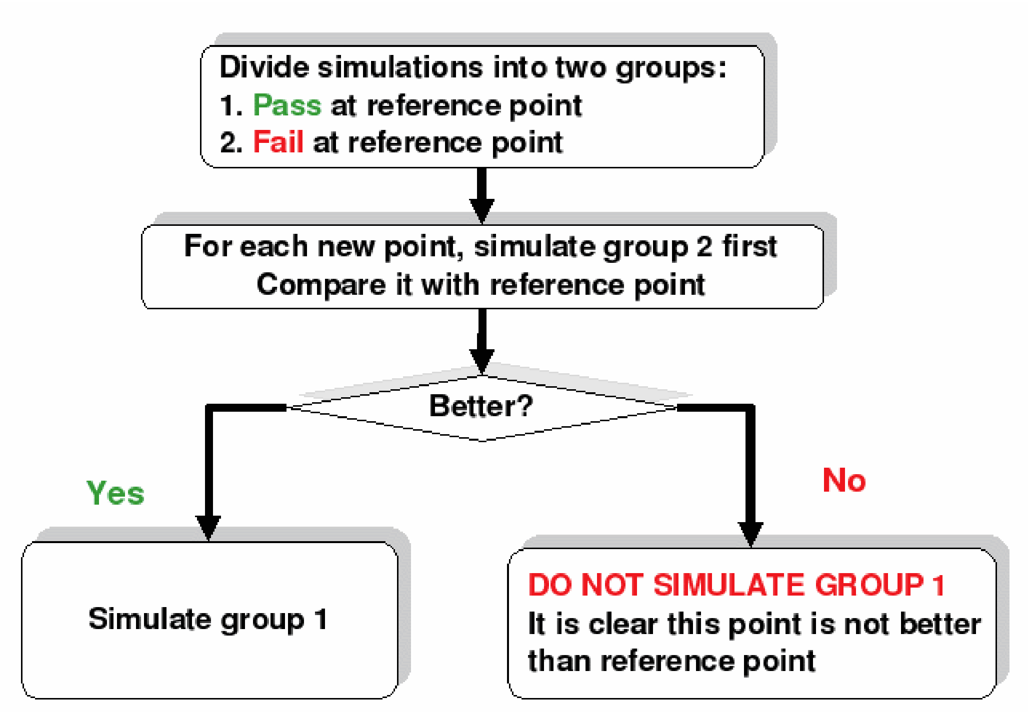
Note: To perform an optimization, ensure that the design meets the following requirements:

- ❑ Specification targets are defined
- ❑ Specification output expressions are scalar results and do not include waveforms.

Understanding Conditional Evaluation

As ADE GXL is running the simulations for each design point, it may encounter a point where the first simulation run is not as good as the current best design point. In this case, ADE GXL will not run any remaining simulations in order to more quickly and effectively reach a solution. This type of evaluation is called conditional evaluation, and may result in only partially-evaluated points.

When sizing a design with conditional evaluation set, ADE GXL follows the following procedure:



The “No” case results in a partially-evaluated point.

Running a Local Optimization

You can perform local optimization by using any of the four algorithms: *Brent-Powell*, *Hooke-Jeeves*, *BFGS*, and *Conjugate Gradient*. You can consider the following points while making a choice of an algorithm to be used to run local optimization:

- The *BFGS* and *Conjugate Gradient* algorithms are gradient-based algorithms. These methods do well when gradient information is available, that is, when specification measurements return more than two or three states and they do not result in evaluation errors.

  BFGS uses second order Hessian Matrix to search local optimum, which is much more efficient than Conjugate Gradient if the performance space is closer to quadratic. Therefore, it is recommended to use the BFGS algorithm in cases where performance gradient can be calculated and design variables are naturally continuous.

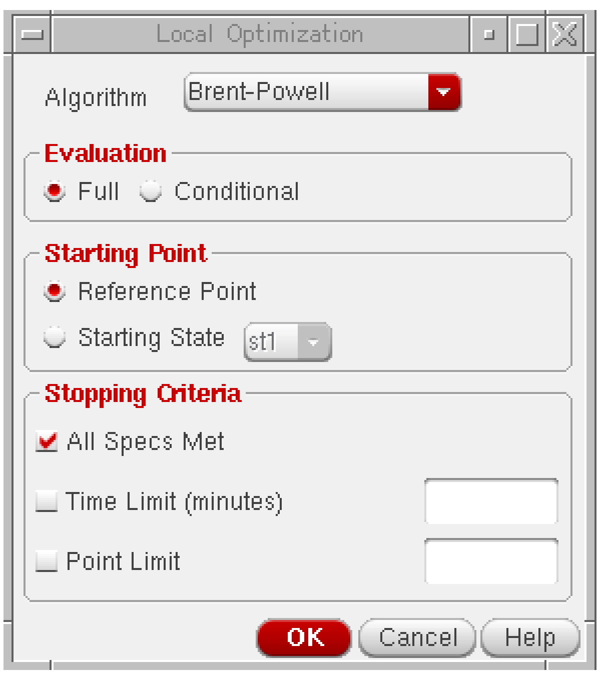
- The *Brent-Powell* algorithm searches in a fine grid around the starting point. Use the *Brent-Powell* option if your reference point is already close (e.g. only a few specifications not met). With this type, ADE GXL should find the local minimum around the selected point, but it may also quit once that local minimum is found and then miss a more optimal solution in the local design space. Because it only runs one point at a time, ADE GXL is slow with this option, and it runs the least number of points on average.

- The type *Hooke-Jeeves* algorithm searches in bigger steps, but it can miss a local minimum that might have been found by the *fine* option. Use the *Hooke-Jeeves* option if your time is limited and the starting point is not as close. With this option, ADE GXL is less likely to quit prematurely, but it also may miss a local minimum that the *Brent-Powell* option would have found. With the *Hooke-Jeeves* option, ADE GXL runs more points, but it is faster on multiple machines than when running with the *Brent-Powell* option because of parallel points.

To run local optimization:

1. From the *Select a Run Mode* drop-down list on the Run toolbar, choose *Local Optimization*.
2. Click *Simulation Options*  to specify optimization options.

   The Local Optimization Options form appears.



3. From the *Algorithm* drop-down list, select the algorithm for local optimization.

   The following algorithms are supported:

   - Conjugate Gradient

- ❑ Brent-Powell
- ❑ Hooke-Jeeves
- ❑ BFGS

4. Select an evaluation type by selecting one of the following radio buttons:
   - ❑ Full
   - ❑ Conditional

   For more information on conditional evaluation, see Understanding Conditional Evaluation on page 132.

5. In the *Starting Point* panel, select one of the following options to specify a starting point for the simulation run:
   - ❑ *Reference Point*—Select this option if you have created a reference point and want to use that point as the starting point for the run.
   - ❑ *Starting State*—Select this option if you have created a setup state and want to use that as the starting point for the run. To use the setup state as the starting point for the optimization, select a setup state that defines a set of fixed values for every global variable or parameter that defines a range of values in the active setup.

   Note: You must have a reference point or a state available to use this option.

6. If desired, select the criteria for the duration of time for which the local optimization should run. You can select one or more of the following:
   - ❑ To run only until all goals are met, select the *All Specs Met* check box.
   - ❑ To set a time limit for the run, select the *Time Limit* check box and enter a value in minutes.
   - ❑ To set a limit in the number of points run, select the *Point Limit* check box and enter the number of points.

7. Click *OK*.

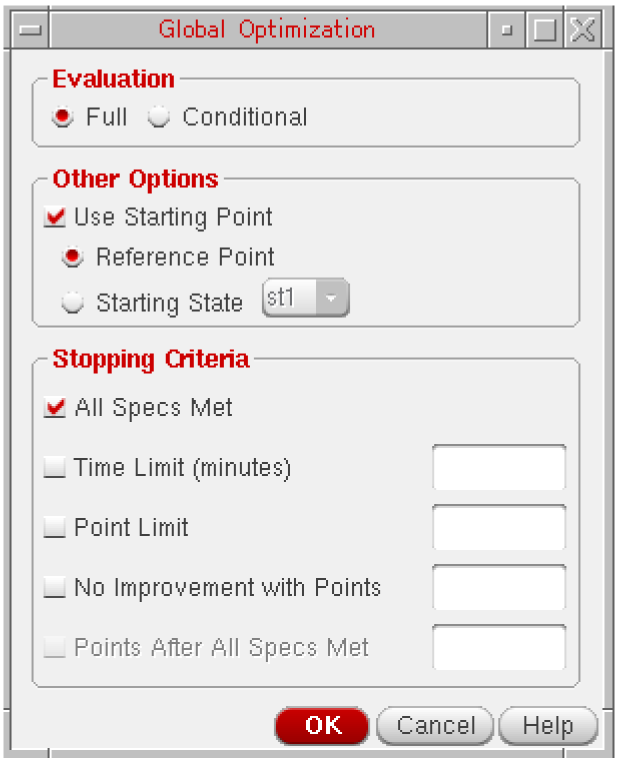
8. Click the *Run Simulation*  button on the Run toolbar to optimize the circuit.

Running a Global Optimization

To run a global optimization:

1. From the *Select a Run Mode* drop-down list on the Run toolbar, choose *Global Optimization*.
2. Click *Simulation Options*  to specify optimization options.

   The Global Optimization form appears.



3. Select an evaluation type by selecting one of the following radio buttons:
   - ❑ Full
   - ❑ Conditional

   For more information on conditional evaluation, see Viewing the Variable Data from Optimization Results.

4. If you want to specify a starting point for the simulation run, select the *Use Starting Point* check box in the *Other Options* panel. After you select this check box, you need to select one of the following options:

- ❑ *Reference Point*—Select this option if you have created a reference point and want to use that point as the starting point for the run.
- ❑ *Starting State*—Select this option if you have created a setup state and want to use that as the starting point for the run. To use the setup state as the starting point for the optimization, select a setup state that defines a set of fixed values for every global variable or parameter that defines a range of values in the active setup.

Note: You must have a reference point or a state available to use this option.

5. If you have created a schematic point or a reference point, and you want to use that point as a starting place for sizing, select the *Use Reference Point as Starting Point* check box.

   You must have a reference point available to utilize this option.

6. If desired, select criteria for the length of time local optimization should run.

   You can select one or more of the following:

   - ❑ To run only until all goals are met, select the *All Specs Met* check box.
   - ❑ To set a time limit for the run, select the *Time Limit* check box and enter a value in minutes.
   - ❑ To set a limit in the number of points run, select the *Point Limit* check box and enter the number of points.
   - ❑ To stop sizing when no improvement is seen for a certain number of points, select the *No Improvement with Points* check box and enter the number of points.
   - ❑ To continue exploring the design space for a better solution even all specifications are met, select the *Points After All Specs Met* check box and enter the number of points to explore.

Tip

You cannot select both the *All Specs Met* and *Points After All Specs Met* check boxes, as these two options are mutually exclusive.

7. Click *OK*.
8. Click the *Run Simulation* button on the Run toolbar to optimize the circuit.

Viewing the Variable Data from Optimization Results

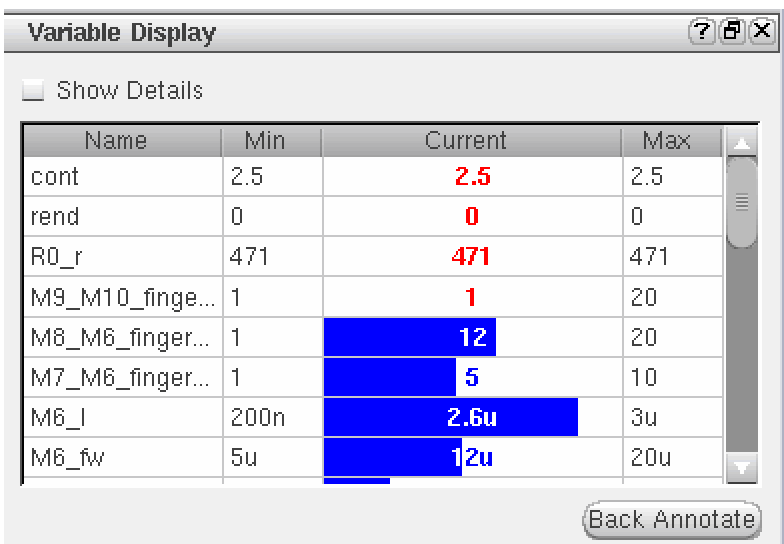
The Variable Display Assistant displays, for a selected design point, the minimum and maximum values for each variable. During optimization, this assistant shows the values for the current best point found, while after optimization, the assistant displays the values for the best point found during the optimization process.

In addition, ADE GXL also displays the current variable values for the selected point, and, through a status bar, shows how far from the minimum and maximum values the current value is. A min or max value in **red** indicates that the current value is pushing the minimum or maximum and may need adjustment for synthesis to be completely successful.

To display the Variable Display Assistant:

➤ Choose *Window – Assistants – Variable Display*.

The Variable Display Assistant appears.



Tip

You can view more details about the variable, including the lib, cell, and view, by selecting the *More Details* check box.

Manual Tuning

While parameterizing a design, you can tune your design by varying the values of parameters, running multiple simulations and then comparing results. You can do this by using the Manual Tuning run mode.

Some of the important points related to the Manual Tuning run mode are as follows:

- You run the Single Run, Sweeps and Corners, Monte Carlo, Global Optimization, and Local Optimization run modes multiple times.

  Currently, the Manual Tuning run mode is not supported in the Improve Yield, Sensitivity Analysis, High Yield Estimation, Create Worst Case Corners, Size Over Corners run modes.

- Every time you tune the design parameters and run a simulation, results are appended to the same history checkpoint. This is as compared to the other run modes in which the results of every simulation run are saved in a new checkpoint.

- You can compare the results data in a single table and plot specifications across design points run in different simulations. Therefore, manual tuning, when used with design parameterization, helps you tune your design without changing the schematic.

To run manual tuning:

1. In the adexl view of your design, from the *Select a Run Mode* drop-down list on the Run toolbar, choose *Manual Tuning*.

2. Click the *Run Simulation* button on the Run toolbar.

   Note: The color of the *Run Simulation* button changes to yellow. This indicates that the Manual Tuning run mode has started.

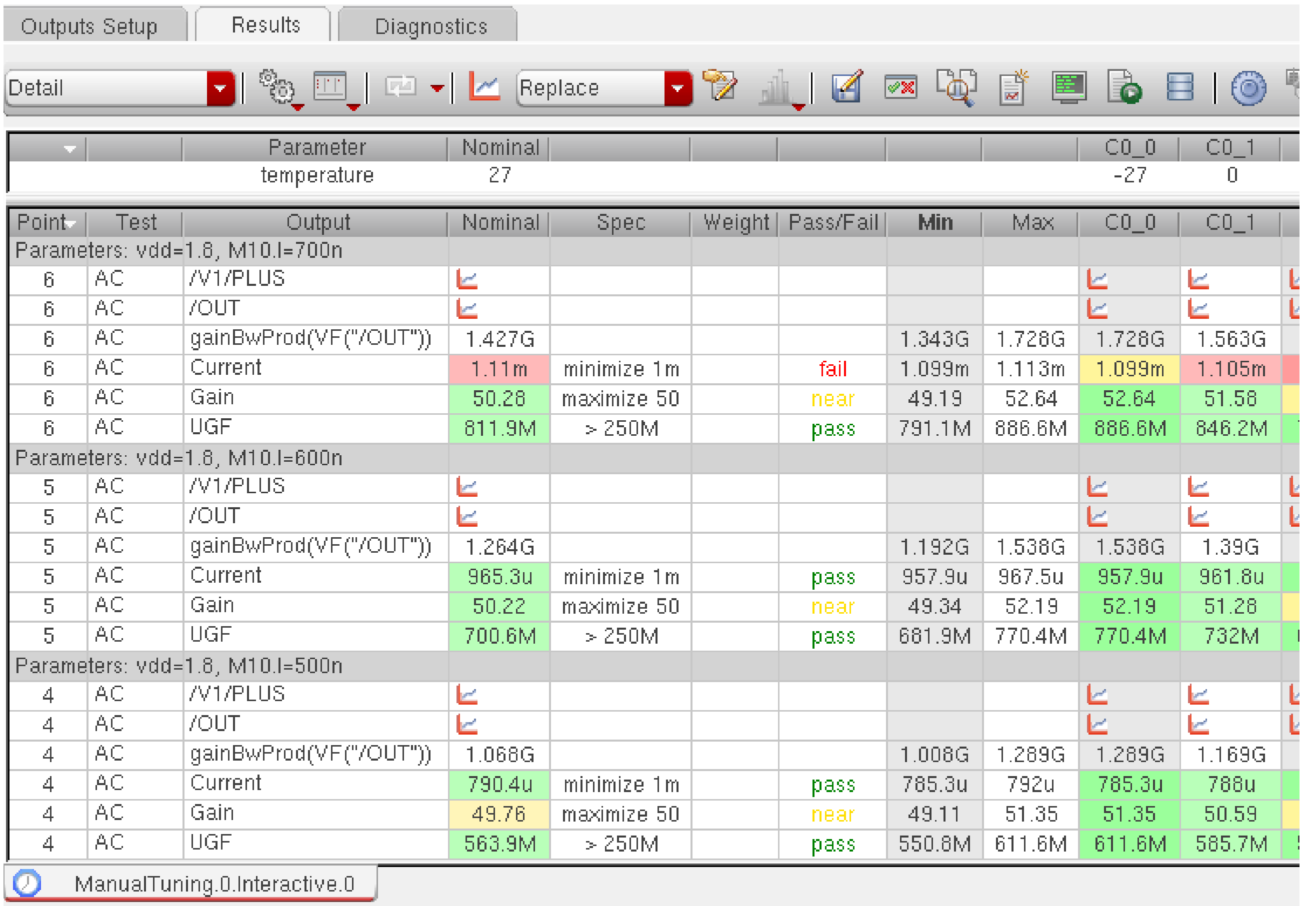
   Now, you can tune or vary the values of design parameters and run multiple simulations to obtain the desired results.

3. In the Data View or the Variables and Parameters assistant, specify values for the design parameters.

4. From the *Select a Run Mode* drop-down list on the Run toolbar, choose *Single Run, Sweeps and Corners*.

5. Click the *Run Simulation* button on the Run toolbar to start the *Single Run, Sweeps and Corners* run.

   The simulation results are displayed on the *Results* tab, as shown below.

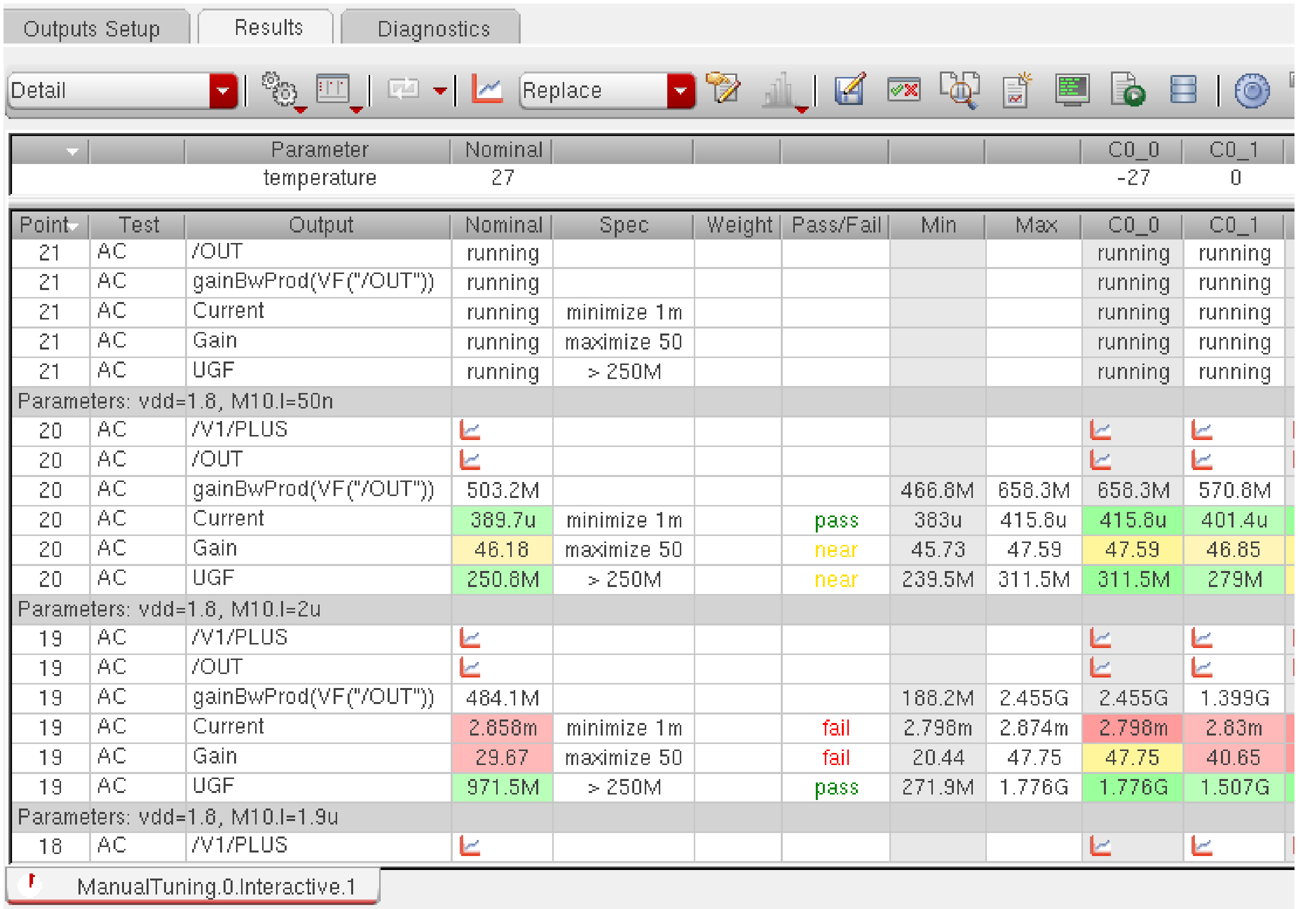


		Parameter	Nominal						C0_0	C0_1
		temperature	27						-27	0
Point	**Test**	**Output**	**Nominal**	**Spec**	**Weight**	**Pass/Fail**	**Min**	**Max**	**C0_0**	**C0_1**
Parameters: vdd=1.8, M10.l=700n										
6	AC	/V1/PLUS								
6	AC	/OUT								
6	AC	gainBwProd(VF("/OUT"))	1.427G				1.343G	1.728G	1.728G	1.563G
6	AC	Current	1.11m	minimize 1m		fail	1.099m	1.113m	1.099m	1.105m
6	AC	Gain	50.28	maximize 50		near	49.19	52.64	52.64	51.58
6	AC	UGF	811.9M	> 250M		pass	791.1M	886.6M	886.6M	846.2M
Parameters: vdd=1.8, M10.l=600n										
5	AC	/V1/PLUS								
5	AC	/OUT								
5	AC	gainBwProd(VF("/OUT"))	1.264G				1.192G	1.538G	1.538G	1.39G
5	AC	Current	965.3u	minimize 1m		pass	957.9u	967.5u	957.9u	961.8u
5	AC	Gain	50.22	maximize 50		near	49.34	52.19	52.19	51.28
5	AC	UGF	700.6M	> 250M		pass	681.9M	770.4M	770.4M	732M
Parameters: vdd=1.8, M10.l=500n										
4	AC	/V1/PLUS								
4	AC	/OUT								
4	AC	gainBwProd(VF("/OUT"))	1.068G				1.008G	1.289G	1.289G	1.169G
4	AC	Current	790.4u	minimize 1m		pass	785.3u	792u	785.3u	788u
4	AC	Gain	49.76	maximize 50		near	49.11	51.35	51.35	50.59
4	AC	UGF	563.9M	> 250M		pass	550.8M	611.6M	611.6M	585.7M

Note: The default name of the Results tab indicates the interactive run number in the Manual Tuning run mode. For example, in the figure shown above, the first single, sweep, corner run in the manual tuning run mode is named as `ManualTuning.0.Interactive.0`.

6. If required, change the value of parameters to tune your design and again click the *Run Simulation* button on the Run toolbar to start a new *Single Run, Sweeps and Corners* simulation run.

Note: The results of the subsequent simulation runs are appended on top of the Results tab already open for the previous run, as shown below.



Parameter	Nominal	C0_0	C0_1
temperature	27	-27	0

Point	Test	Output	Nominal	Spec	Weight	Pass/Fail	Min	Max	C0_0	C0_1
21	AC	/OUT	running						running	running
21	AC	gainBwProd(VF("/OUT"))	running						running	running
21	AC	Current	running	minimize 1m					running	running
21	AC	Gain	running	maximize 50					running	running
21	AC	UGF	running	> 250M					running	running
Parameters: vdd=1.8, M10.l=50n										
20	AC	/V1/PLUS								
20	AC	/OUT								
20	AC	gainBwProd(VF("/OUT"))	503.2M				466.8M	658.3M	658.3M	570.8M
20	AC	Current	389.7u	minimize 1m		pass	383u	415.8u	415.8u	401.4u
20	AC	Gain	46.18	maximize 50		near	45.73	47.59	47.59	46.85
20	AC	UGF	250.8M	> 250M		near	239.5M	311.5M	311.5M	279M
Parameters: vdd=1.8, M10.l=2u										
19	AC	/V1/PLUS								
19	AC	/OUT								
19	AC	gainBwProd(VF("/OUT"))	484.1M				188.2M	2.455G	2.455G	1.399G
19	AC	Current	2.858m	minimize 1m		fail	2.798m	2.874m	2.798m	2.83m
19	AC	Gain	29.67	maximize 50		fail	20.44	47.75	47.75	40.65
19	AC	UGF	971.5M	> 250M		pass	271.9M	1.776G	1.776G	1.507G
Parameters: vdd=1.8, M10.l=1.9u										
18	AC	/V1/PLUS								

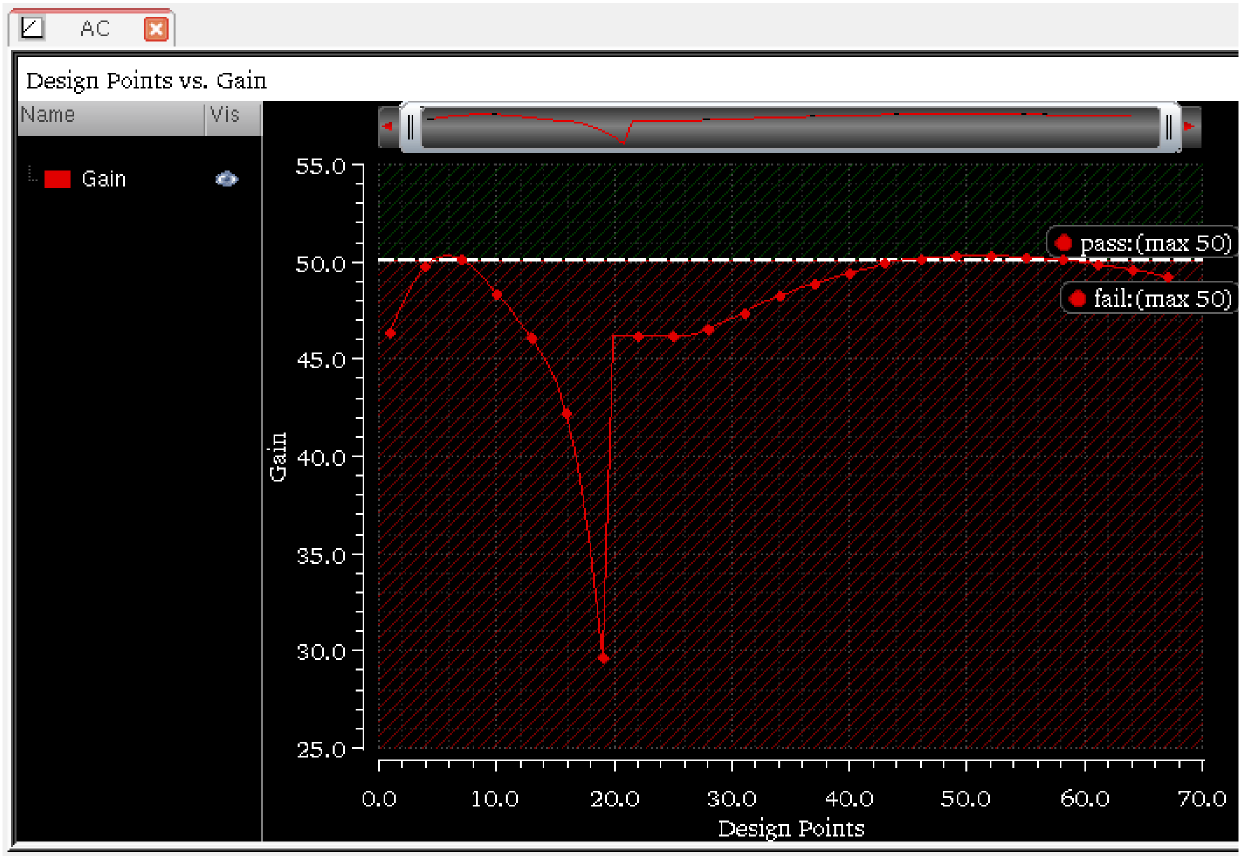
In the other run modes, results of every next simulation run are displayed on a new tab. Also, note that the name of the Results tab has also changed to `ManualTuning.0.Interactive.1`.

At the end of the second simulation run, the Results tab contains all the design points run during the `Interactive.0` and `Interactive.1` runs.

7. Compare the results of the two simulation runs.

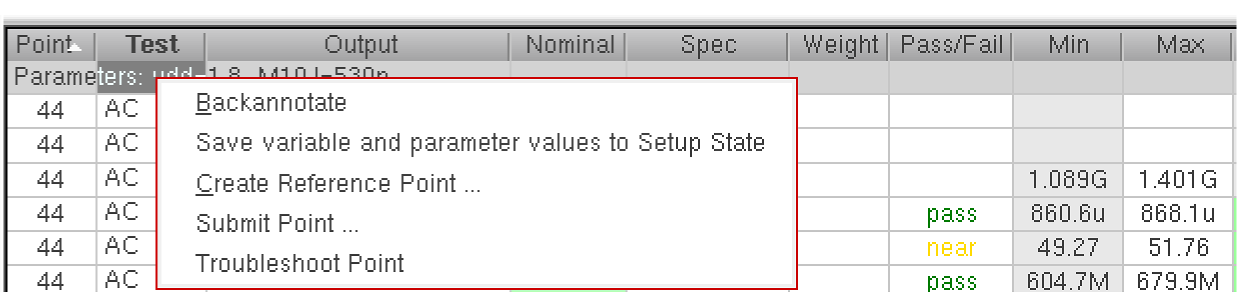
   You can also plot the results of all the simulations run in a manual tuning run. For that, in the Results tab, right-click on a specification and choose *Plot Across Design Points*.

   Note that when you plot results across different design points, data is plotted for all the points of the two runs, as shown below.



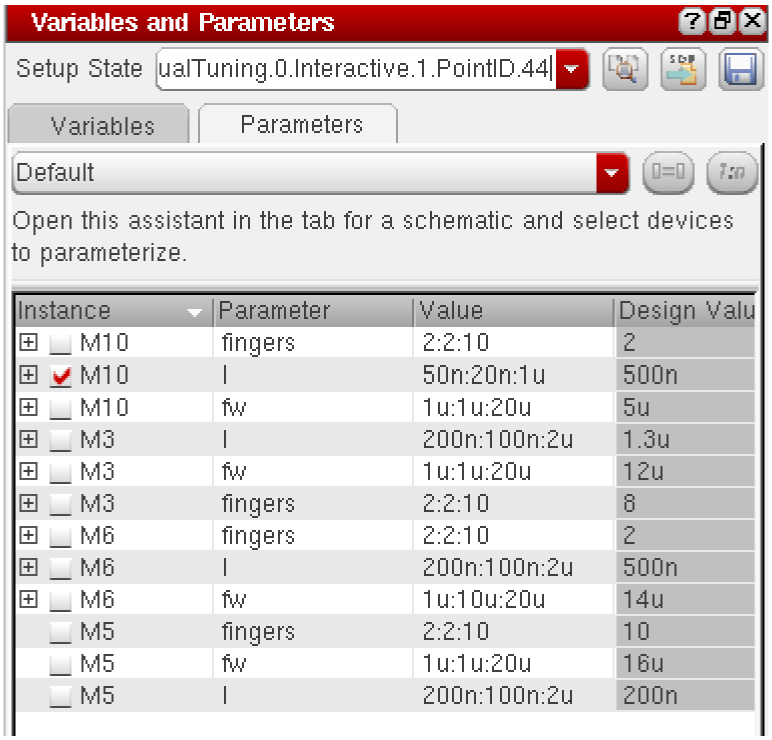
8. (Optional) If the desired results are not obtained, you might choose to further tune the design parameters and run more simulations till you get the best design point.

9. After the desired results are obtained, sort the design points to view the best or the worst point. If the best point meets the specifications, you can save the values used for that point so that they can be reused later. To do this, right-click on the gray row on top of that design point and choose *Save variable and parameter values to Setup State*.



ADE XL creates a state by using the names of the simulation run and the design point ID. For example, for the design point 44 shown above, the tool creates a state named

`ManualTuning.0.Interactive.1.PointID.44` and saves the values of variables and design parameters in that state.



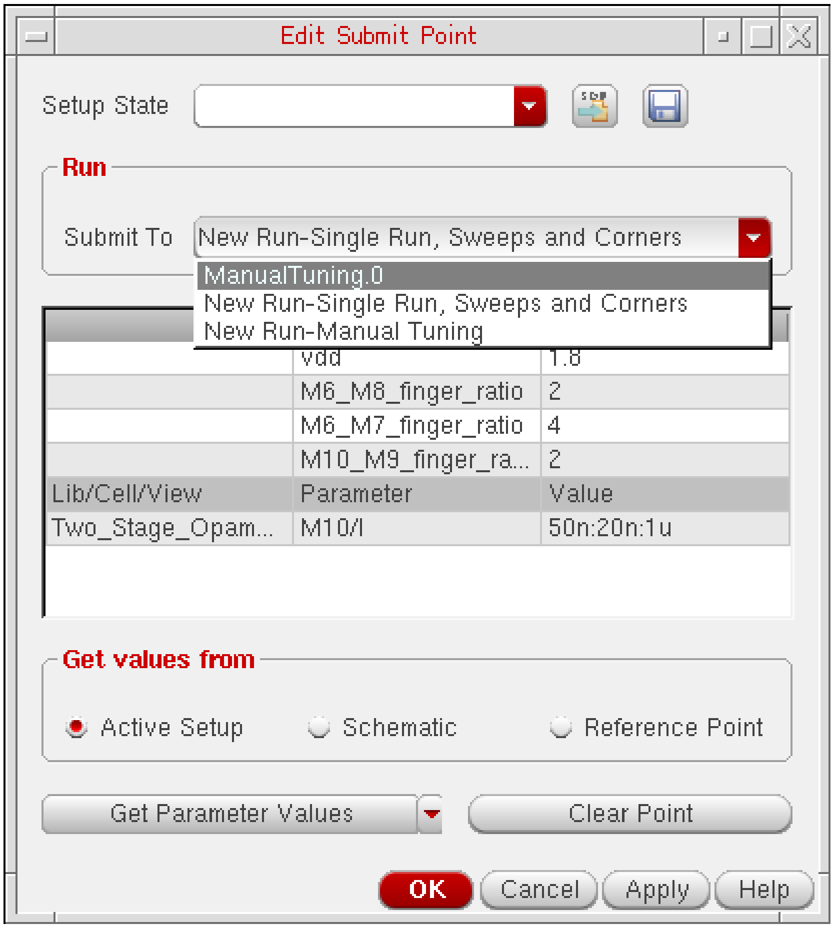
Instance	Parameter	Value	Design Valu
M10	fingers	2:2:10	2
M10	l	50n:20n:1u	500n
M10	fw	1u:1u:20u	5u
M3	l	200n:100n:2u	1.3u
M3	fw	1u:1u:20u	12u
M3	fingers	2:2:10	8
M6	fingers	2:2:10	2
M6	l	200n:100n:2u	500n
M6	fw	1u:10u:20u	14u
M5	fingers	2:2:10	10
M5	fw	1u:1u:20u	16u
M5	l	200n:100n:2u	200n

Note: You can use this saved state in the future to load the values of variables and parameters. For more details, see Working with Global Variables in *Analog Design Environment XL User Guide*.

10. Click *Stop Simulation* on the *Run* toolbar to exit the Manual Tuning run mode.

Important

After exiting the Manual Tuning run mode, you can continue to run more simulations in a previous run by submitting more points to that run. To do this, you can open the **Edit Submit Point** form and submit a point to a previous manual tuning run listed in the *Submit To* drop-down list.

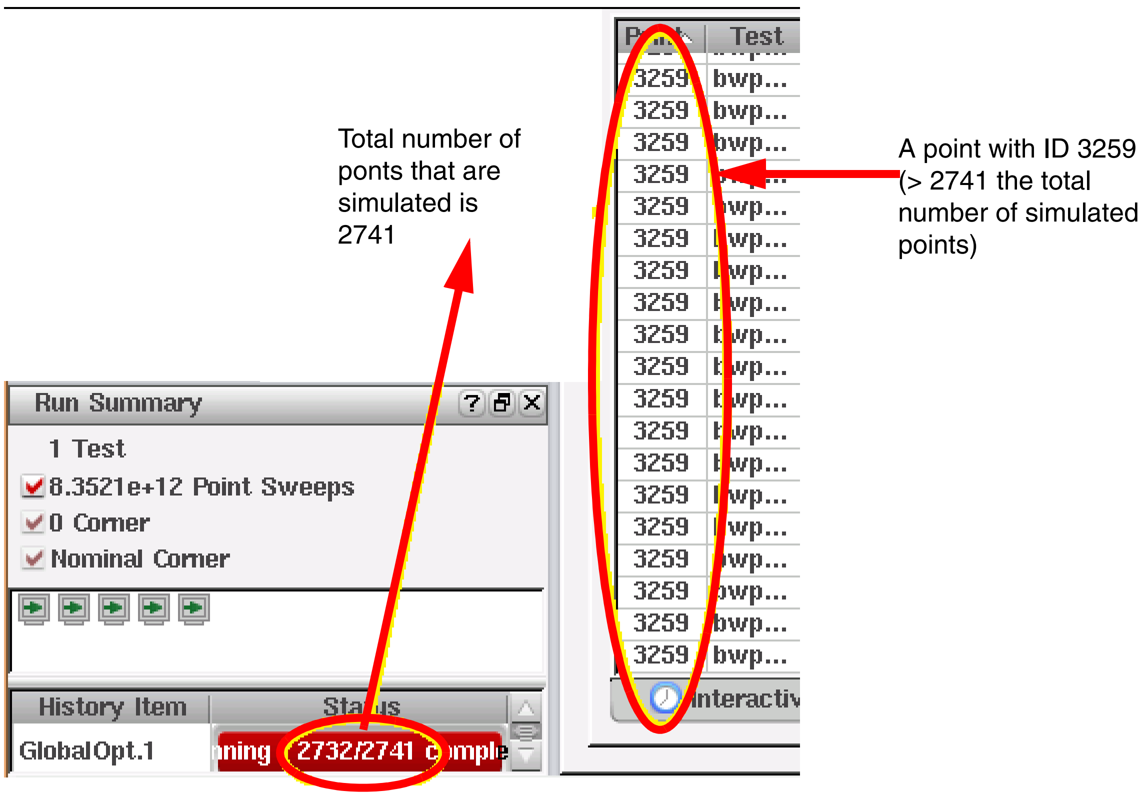


For example, if you choose `ManualTuning.0` in the form shown above and click *OK*, the ManualTuning.0 run is started again, results are reloaded in the Results tab, and the new design points are appended to the existing results.

Important Notes

- If you do not exit the Manual Tuning run mode and close the ADE XL GUI, the currently running Manual Tuning is not stopped. When you open a new ADE XL session, the Run Simulation button is yellow in color and you can continue with the same Manual Tuning run that was running earlier. If you want that the currently running Manual Tuning run should stop when the ADE XL GUI is closed, set the stopManualTuningOnSessionExit environment variable to `t`.
- Since the Manual Tuning results are saved to a single database, it is not possible to delete a specific child history or the simulation data for a specific child history. Therefore, on the Data View pane, the right-click menu commands, *Delete* and *Delete Simulation Data*, are not available for the child histories of a Manual Tuning run.

- During optimization, the optimizer assigns a unique point ID numbers to all the design points. The optimizer may visit the design point or set of design parameters twice or more. In this case, the optimizer assigns a new ID number to the design point everytime when it is visited. This is called a cache-hit because the results for this point already exist in the database and simulator does not need to re-simulate this design point. When the design points are visited more than once, you may notice a greater point ID number than the total number of unique design points that are simulated. For example, consider the scenerio below:



Sizing Over Corners

While you can always size with all corners enabled using local or global optimization, it may not be efficient to do so with a large number of corners.

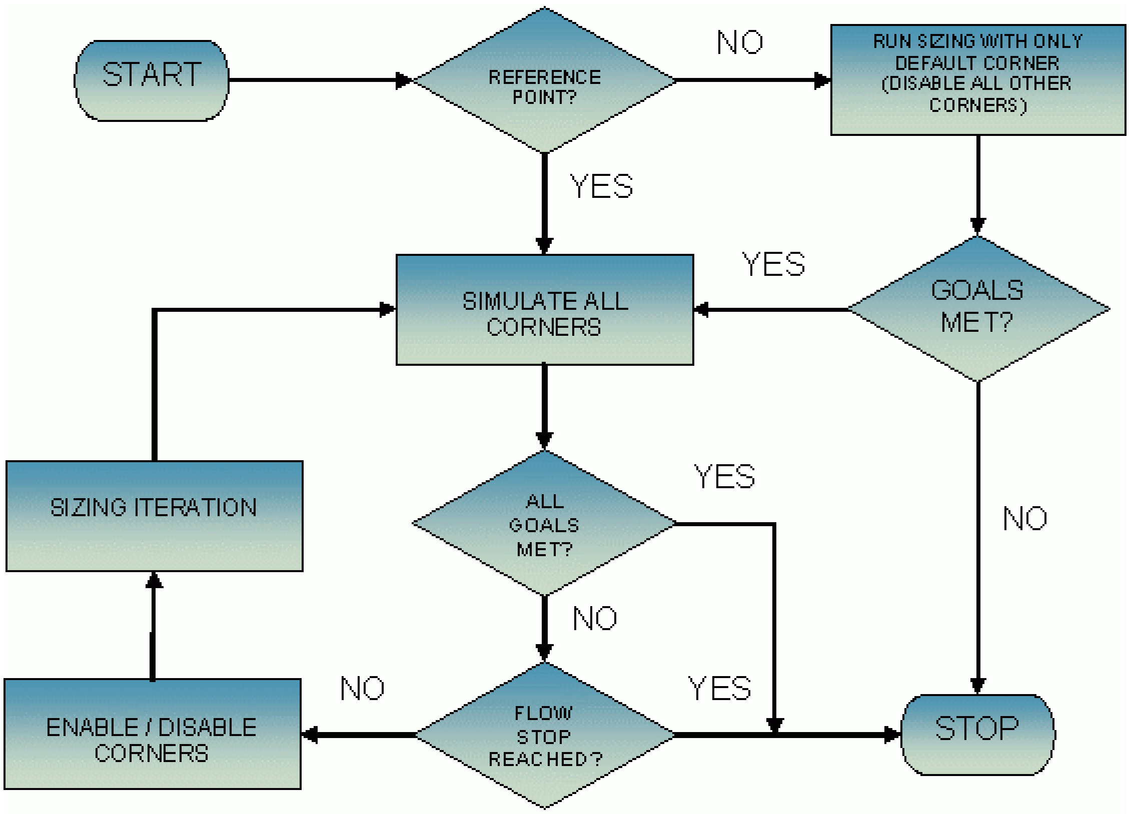
The Sizing Over Corners feature is an intelligent algorithm designed to optimize the test benches over a large number of corners. It does this by running a single point over all user-defined corners, determining the worst-case corners, using the worst-case corners for optimization and improving those corners on each iteration.

You can either provide a starting point, or ADE GXL performs an initial global optimization on the nominal corner and uses its best point as the starting point for the run. ADE GXL simulates all corners to determine the worst case performance corners for each goal. ADE GXL then resizes, taking into account the identified worst case performance corners. This process repeats until the stopping criteria for your size over corner run are met—either until all goals are met, or until it has run through the specified number of iterations or time limit.

Note: If the run stops due to the number of iterations or time limit being met, a final corner simulation is run if the last history children item for the run is related to sizing/optimization.

Figure 4-1 illustrates the sizing over corners flow.

Figure 4-1 Flow for Sizing Over Corners



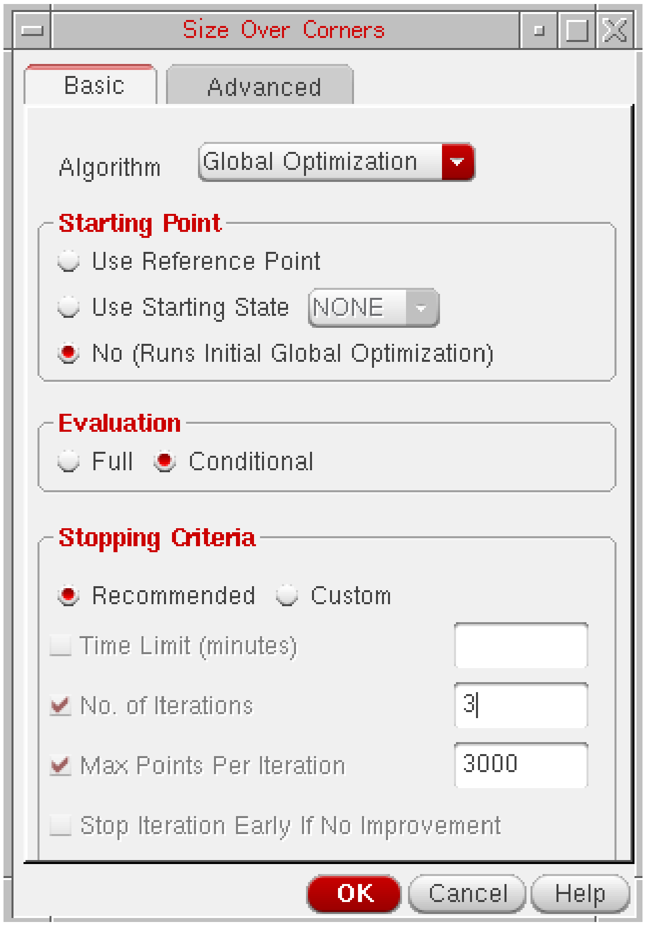
To size over corners, do the following:

1. From the *Run* menu, select *Size Over Corners*.

Tip

Alternatively, select *Size Over Corners* in the *Select a Run Mode* drop-down list on the Run toolbar, then click the *Simulation Options* button on the Run toolbar.

The Size Over Corners form appears.



2. From the *Algorithm* drop-down list on the Basic tab, select the algorithm for sizing over corners.

The default algorithm is *Global Optimization*.

3. In the *Starting Point* panel, select one of the following options to specify a starting point for the simulation run:

- *Use Reference Point*—Select this option if you have created a reference point and want to use that point as the starting point for the run.

- *Use Starting State*—Select this option if you have created a setup state and want to use that as the starting point for the run. To use the setup state as the starting point for the optimization, select a setup state that defines a set of fixed values for every global variable or parameter that defines a range of values in the active setup.

  You must have a reference point or a state available to use this option.

- ❑ *No (Runs Initial Global Optimization)*—Select this option if you don't have a reference point, or do not want to use the reference point as the starting point for sizing. When this option is selected, ADE GXL performs an initial global optimization on the nominal corner and uses its best point as the starting point for the run. If you select *No (Runs Initial Global Optimization)*, ensure that:
- ❑ The nominal corner is not disabled in the Run Summary pane.

  For more information, see the Disabling and Enabling the Nominal Corner.
- ❑ For tests that are enabled in the Data View pane, the nominal corner is not disabled in the Corners Setup form.

  For more information, see Disabling and Enabling the Nominal Corner for Specific Tests.

4. In the *Evaluation* group box, select one of the following evaluation types for the sizing run:
   - ❑ Full
   - ❑ Conditional

   For more information on conditional evaluation, see Understanding Conditional Evaluation on page 132.

5. Select the *Recommended* option under *Stopping Criteria* to use the recommended criteria for the length of time size over corners should run. The recommended criteria are:
   - ❑ Three sizing iterations
   - ❑ 3000 points run per iteration

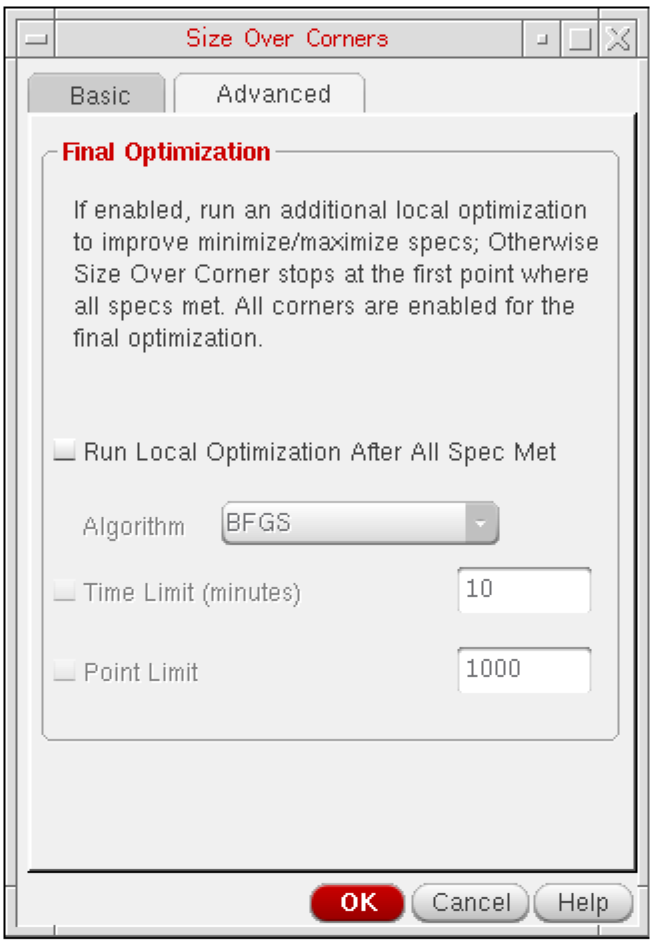
   If you want to modify these defaults, select the *Custom* option, then select one or more of the following stopping criteria:
   - ❑ To set a time limit for the run, select the *Time Limit (minutes)* check box and enter a value in minutes.
   - ❑ To specify the number of sizing iterations, select the *No. of Iterations* check box and enter the number in the field.
   - ❑ To specify the maximum number of points processed per iteration, select the *Max Points per Iteration* check box and enter the number of points in the field.
   - ❑ To stop the process early if the sizing results in no improvement, select the *Stop Iteration Early if No Improvement* check box. This option is applied to each optimization iteration. This stopping criteria is similar to the *No Improvement with Points* stopping criteria for the global or local optimization run modes for which the

user also specifies the number of points. In the case of iterative run modes, the number of points is calculated as `Max Points per Iteration / 3`.

Note: If all goals are met prior to the stopping criteria being reached, ADE GXL will stop the sizing run.

6. (Optional) By default, the Size Over Corners run is automatically stopped when all the goals are met. If the test outputs have minimize or maximize specs defined for them, you can choose to automatically run the local optimization after the best design point is found. This helps in getting better results for these types of specifications.

   For this, open the Advanced tab.



   Select the *Run Local Optimization After All Spec Met* option. This specifies that local optimization needs to be run after all the specs are met. The other options on this tab are used by the local optimization run. If required, change the default values.

7. Click *OK* to save the changes and close the Size Over Corners form.

8. Click the *Run Simulation* button on the Run toolbar.

ADE GXL runs through iterations of sizing and corner sweeping until your stopping criteria are met.

Running Feasibility Analysis

Feasibility analysis allows you to quickly verify that the operating region specifications can be satisfied. It does this by running optimization based on the operating region specifications set up for your circuit to evaluate if any design point passes the operating region specifications.

If feasibility analysis identifies design points that pass all operating region specifications, you can continue with tuning the design over other specifications or run optimization. For more information about running optimization, see Running Optimization on page 132. If no design point passes all operating region specifications, it does not mean that there are absolutely no feasible points in the solution space. In this case, you may change the design constraints or the circuit topology and then check if any design point passes all operating region specifications.

Note: Feasibility analysis evaluates only the operating region specifications. The tests, analyses and specifications that are not related to operating region specifications are not evaluated when you run feasibility analysis.

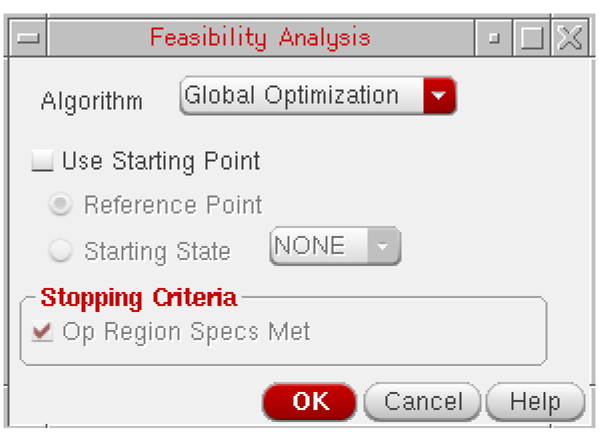
To run feasibility analysis, do the following:

1. Specify operating region specifications for your circuit. For more information, see Running Optimization on page 132.
2. From the *Run* menu, select *Feasibility Analysis*.

Tip

Alternatively, select *Feasibility Analysis* in the *Select a Run Mode* drop-down list on the Run toolbar, then click the *Simulation Options* button on the Run toolbar.

The Feasibility Analysis form appears.



3. From the *Algorithm* drop-down list, select the algorithm for optimizing the design to meet the operating region specifications.

   The following algorithms are supported:

   - Global Optimization
   - Conjugate Gradient
   - Brent-Powell
   - Hooke-Jeeves
   - BFGS

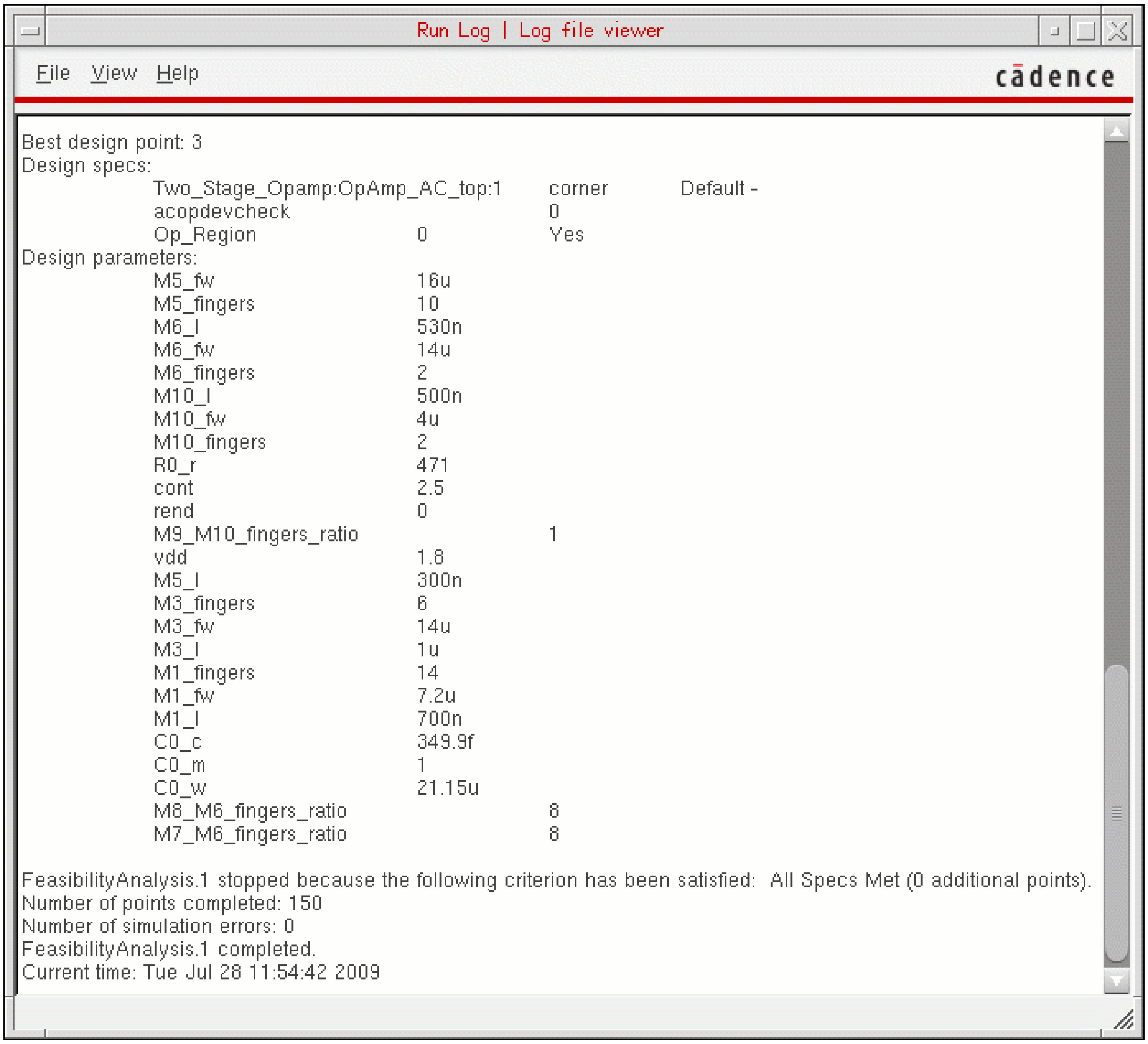
4. Select the *Use Reference Point as Starting Point* check box to specify a starting point for the simulation run. Select one of the following options:

   - *Use Reference Point*—Select this option if you have created a reference point and want to use that point as the starting point for the run.
   - *Use Starting State*—Select this option if you have created a setup state and want to use that as the starting point for the run. To use the setup state as the starting point for the optimization, select a setup state that defines a set of fixed values for every global variable or parameter that defines a range of values in the active setup. You must have a reference point or a state available to use this option. Note the following:
     - You must have a reference point when the specified algorithm is *Conjugate Gradient*, *Brent-Powell* or *Hooke-Jeeves*.
     - Selecting the *Use Reference Point as Starting Point* check box is optional when the specified algorithm is Global Optimization.

5. Click *OK* to close the Feasibility Analysis form and start the feasibility analysis run.

   Note: If you opened the Feasibility Analysis form by clicking the *Simulation Options* button on the Run toolbar, click the *Run Simulation* button on the Run toolbar to start the feasibility analysis run.

Information regarding design points that pass all operating region specifications are displayed in the Run Log | Log File Viewer form.



Important

A feasibility analysis run stops only when all operating region specifications are met. If you want to stop the run before this is achieved, click the *Stop Simulation* button on the *Run* toolbar.

Improving the Yield

The *Improve Yield* command can be used to return a design point that meets all corners and has the highest possible yield.

This command runs iterations of sizing and Monte Carlo analysis to arrive at a solution. When you start Improve Yield, ADE GXL first generates the statistical corners, then, as the run progresses, evaluates points on a subset of those corners. Promising points are then evaluated on a larger set of corners. Eventually ADE GXL arrives at the best point -- one that has been evaluated at all statistical corners and has the highest possible yield.

Also available are a number of stopping criteria, including time and points limits. Once ADE GXL hits any of the specified options, it will end the improvement process.

Important

Improve Yield, like *Monte Carlo Sampling*, is available only for the Spectre circuit simulator.

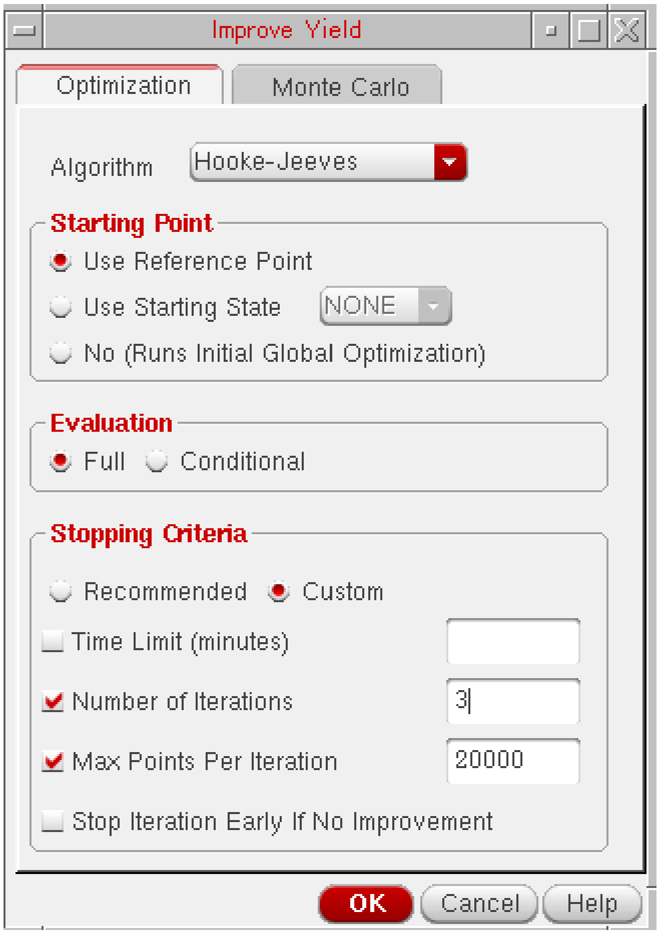
Note that your design must include devices or device models for which you have specified statistically varying parameter values. You must have one or more specs defined and enabled. You must specify either global (process) or mismatch (per-instance) variations or both. You can also specify correlation information. After simulating, you can select the yield view to view mean and standard deviation information. For more information, see "Performing Monte Carlo Analysis" in the *Virtuoso Analog Design Environment XL User Guide*.

Note: For information about specifying parameter distributions for Spectre circuit simulation, see Specifying Parameter Distributions Using Statistics Blocks in the Analyses chapter of the *Spectre Circuit Simulator User Guide*.

To improve the yield:

1. From the *Select a Run Mode* drop-down list on the Run toolbar, choose *Improve Yield*.
2. Click  to specify improve yield options.

The Improve Yield options form appears.



3. On the Optimization tab, select the optimization algorithm from the *Algorithm* drop-down list.

4. In the *Starting Point* panel, select one of the following options to specify a starting point for the simulation run:

- *Use Reference Point*—Select this option if you have created a reference point and want to use that point as the starting point for the run.
- *Use Starting State*—Select this option if you have created a setup state and want to use that as the starting point for the run. To use the setup state as the starting point for the optimization, select a setup state that defines a set of fixed values for every global variable or parameter that defines a range of values in the active setup.

  You must have a reference point or a state available to use this option.

- *No (Runs Initial Global Optimization)*—Select this option if you do not have a reference point, or do not want to use the reference point as the starting point for sizing. When this option is selected, ADE GXL performs an initial global optimization on the nominal corner and uses its best point as the starting point for the run. If you select *No (Runs Initial Global Optimization)*, ensure that:
  - The nominal corner is not disabled in the Run Summary pane. For more information, see the Disabling and Enabling the Nominal Corner.
  - For tests that are enabled in the Data View pane, the nominal corner is not disabled in the Corners Setup form.
  - For more information, see Disabling and Enabling the Nominal Corner for Specific Tests.

5. Select an evaluation type by selecting one of the following radio buttons under *Evaluation*:

- Full
- Conditional

For more information on conditional evaluation, see Understanding Conditional Evaluation.

6. Select the *Recommended* check box under *Stopping Criteria* to use the recommended options. The recommended options are:

- Three sizing/Monte Carlo iterations
- 3000 points run per iteration

If you want to modify these defaults, select the *Custom* check box, then one or more of the following stopping criteria:

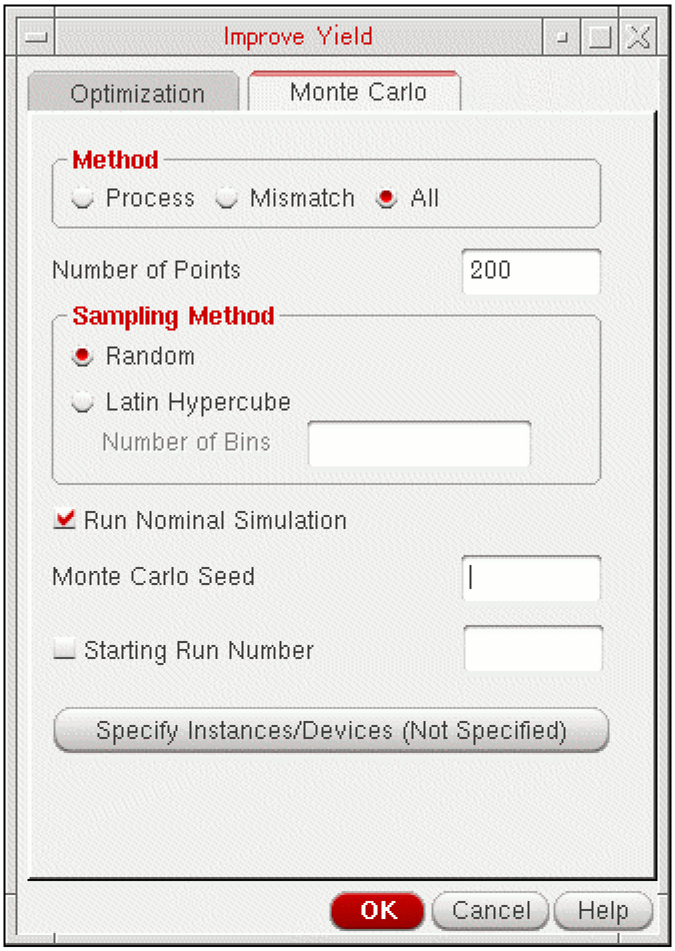
a. To set a time limit for the run, select the *Time Limit (minutes)* check box and enter a value in hours.

b. To specify the number of sizing/Monte Carlo iterations, select the *No. of Iterations* check box and enter the number in the field.

c. If you want to specify the maximum number of points processed per iteration, select the *Max Points per Iteration* check box and enter the number of points in the field.

d. If you want to stop the process early if the sizing results in no improvement, select the *Stop Iteration Early if No Improvement* check box. This option is applied to each optimization iteration. This stopping criteria is similar to the *No Improvement with Points* stopping criteria for the global or local optimization run modes for which the user also specifies the number of points. In the case of iterative run modes, the number of points is calculated as `Max Points per Iteration / 3`.

7. Select the Monte Carlo tab to specify options for Monte Carlo.



8. When you run Improve Yield, you have a choice of varying the process statistical variables, mismatch statistical variables, or both. If you run only one type of statistical variable, the other variables are set to fixed values.

In the *Method* group box, select one of the following statistical variations:

Process	for process statistical variations
Mismatch	for per-instance statistical variations
All	for both process and per-instance statistical variations

Important

You must define your models so that they respond to the statistical variations you choose. You must specify the file containing your models on the Model Library Setup form. For a Spectre circuit simulator example of how to define your models, see "Specifying Parameter Distributions Using Statistics Blocks" in the *Virtuoso Spectre Circuit Simulator User Guide*.

9. In the *Number of Points* field, type the number of Monte Carlo points you want to simulate.

10. In the *Sampling Method* group box, select the statistical sampling method to be used—*Random* or *Latin Hypercube*.

11. If the selected sampling method is *Latin Hypercube*, specify the number of bins (subdivisions) for the *Latin Hypercube* method in the *Number of Bins* field.

Note the following:

- If a number is specified, the number of bins will be the specified number, or *Number of Points* + *Starting Run Number* - `1`, whichever is greater. For example, if the specified number of bins is `90`, the number of points specified in the *Number of Points* field is `100` and the starting run number specified in the *Starting Run Number* field is `6`, the value `105` (100+6-1) is used.
- If no number is specified, a default value of *Number of Points* + *Starting Run Number* - `1` is used. For example, if the number of points specified in the *Number of Points* field is `100` and the starting run number specified in the *Starting Run Number* field is `6`, the default value of `105` (100+6-1) is used.

12. If you want to run a simulation at the reference point prior to beginning the improve yield process, select the *Run Nominal Simulation* check box.

When this option is selected, Spectre will run a simulation at the reference point, and, if this fails, then the sampling process is not initiated and the simulation stops.

13. (Optional) If you want to specify a different seed for the Monte Carlo analysis, select the *Monte Carlo Seed* check box and enter the seed number.

By always specifying the same seed, you can reproduce a previous experiment. If you do not specify a seed, the value `12345` is used.

14. (Optional) If you want to specify a starting run number, select the *Starting Run Number* check box and enter the starting run number.

The starting run number specifies the run that Monte Carlo begins with. By specifying this number, you can reproduce a particular run or sequence of runs from a previous experiment (for example, to examine an earlier case in more detail).

Tip

To reproduce a run or sequence of runs, you need to specify the same value in the Starting Run # and the Monte Carlo Seed fields.

15. By default, mismatch variations are applied to all subcircuit instances in the design. Click the *Specify Instances/Devices* button to specify the sensitive instances and devices you want to either include or exclude for applying mismatch variations. For more information, see the "Including or Excluding Instances and Devices for Applying Mismatch Variations" section in the *Virtuoso Analog Design Environment XL User Guide*.

Note: The N-Sigma tab is not available on the Improve Yield options form now. You can run High Yield Estimation and create statistical corners to optimize your design and to achieve a yield of less than the desired sigma value for the selected specifications. For more details, refer to Creating Statistical Corners From a Worst Case Distance Analysis.

16. In the Run workspace, click the *Run simulation* icon to improve the yield.

ADE GXL begins synthesizing with Monte Carlo analysis.

When the Improve Yield run is finished, the Data View lists the Improve Yield check point. Expanding this check point displays the different runs that make up a full Improve Yield run, including iterations of Optimization and Monte Carlo. You can view the results of any of these runs by right-clicking and choosing *View Results*.

Creating, Viewing, and Modifying Reference Points

You can create a reference point for Improve Yield, Global Optimization, Monte Carlo Sampling, or Sensitivity Analysis from scratch or based on a point in a run. You can also view and modify the current reference point or a reference point from a run.

For more information, see the following topics:

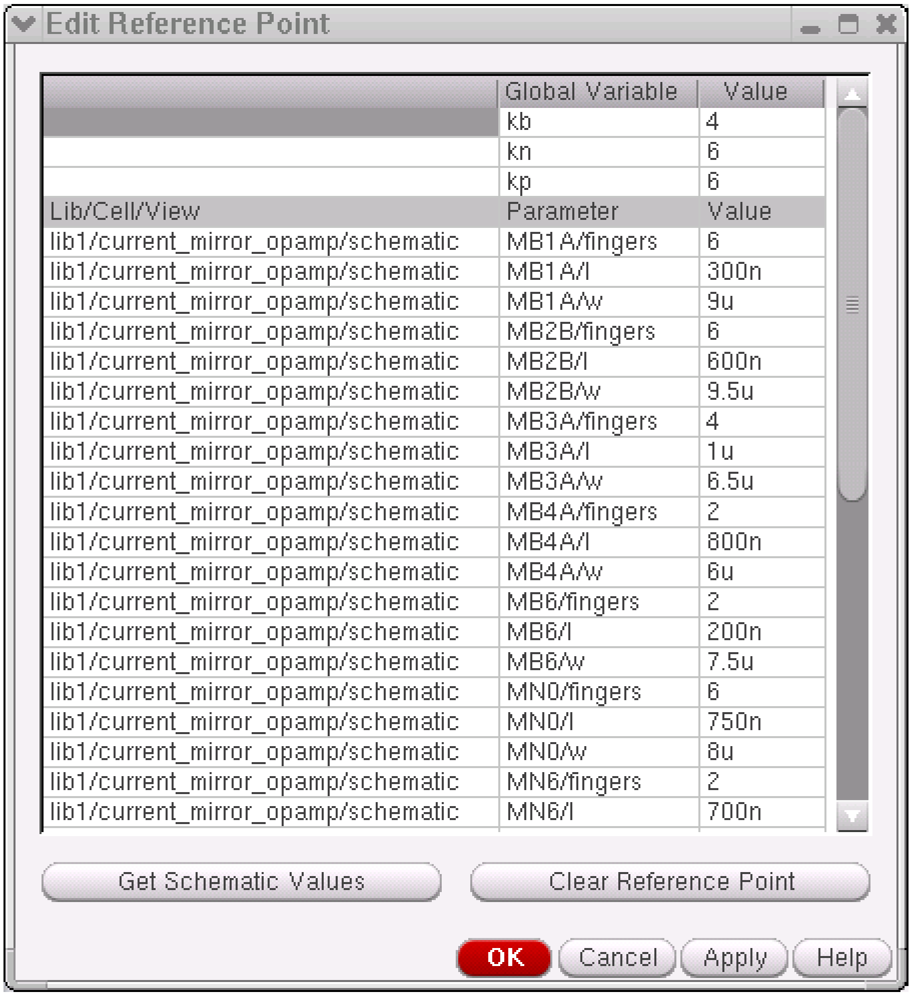
- Creating a Reference Point from Scratch on page 161
- Creating, Viewing, and Modifying a Reference Point from a Run on page 163
- Viewing and Modifying the Current Reference Point on page 165

Creating a Reference Point from Scratch

To create a reference point from scratch:

1. Click the *Edit Reference Point*  icon in the Run Mode tool bar, or choose *Edit Reference Point* from the *Run* menu.

The Edit Reference Point form appears. If you have not previously created a reference point or made any optimization runs, this form will be populated with the schematic values.



2. Add and modify the values as desired:

- ❑ Select the Value field for any parameter and enter a value.
- ❑ To pull in the schematic values do the following:
  - ❍ To pull in the schematic values for specific parameters, select the parameters for which you want the schematic values, click *Get Schematic Values* and choose *Selected Parameters*.
  - ❍ To pull in the schematic values for all the parameters that have no values, click *Get Schematic Values* and choose *Parameters With No Value*.

- To pull in the schematic values for all parameters, click *Get Schematic Values* and choose *All Parameters*.

- Select one or more rows and click *Clear Reference Point* to clear the values.

3. Click *OK* to save your values and dismiss the form.

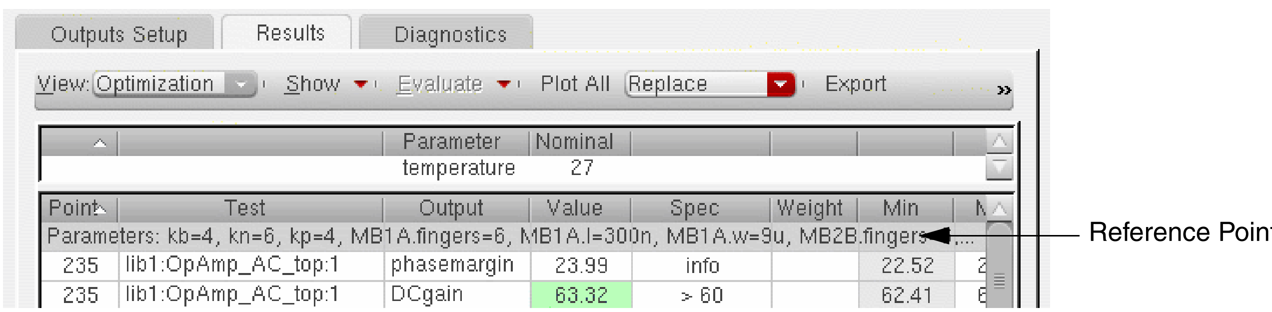
Any reference point you create will remain the active reference point until you create or modify another point.

You can now utilize this point by selecting the *Use Reference Point as Starting Point* check box for Improve Yield or Global Optimization.

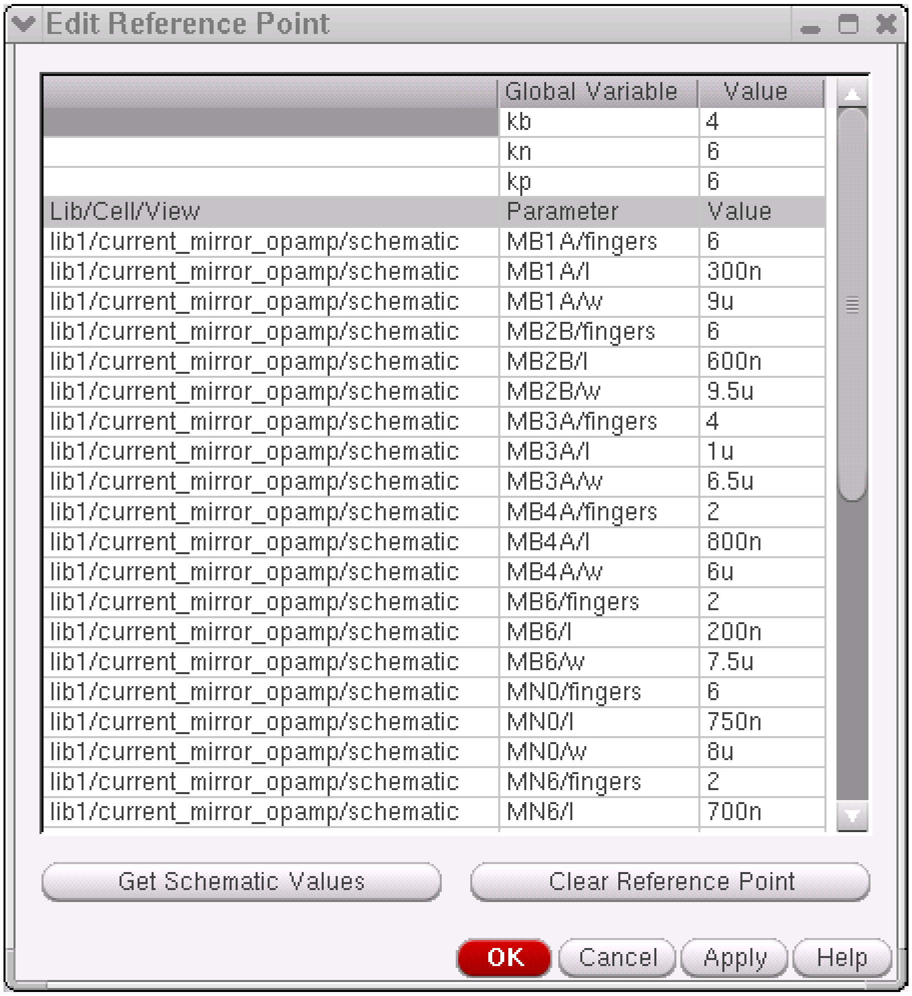
Creating, Viewing, and Modifying a Reference Point from a Run

To create, view, or modify a reference point from a run:

1. Right-click on a point (highlighted in gray) in the Results tab and choose *Create Reference Point*.



The Edit Reference Point form appears.



2. If desired, modify the values as described in step 2 of Creating, Viewing, and Modifying Reference Points:

3. Click *OK* to save your values and dismiss the form.

   If you are merely viewing the point, and haven't made any changes you want to save, click *Cancel*.

   Any reference point you create will remain the active reference point until you create or modify another point.

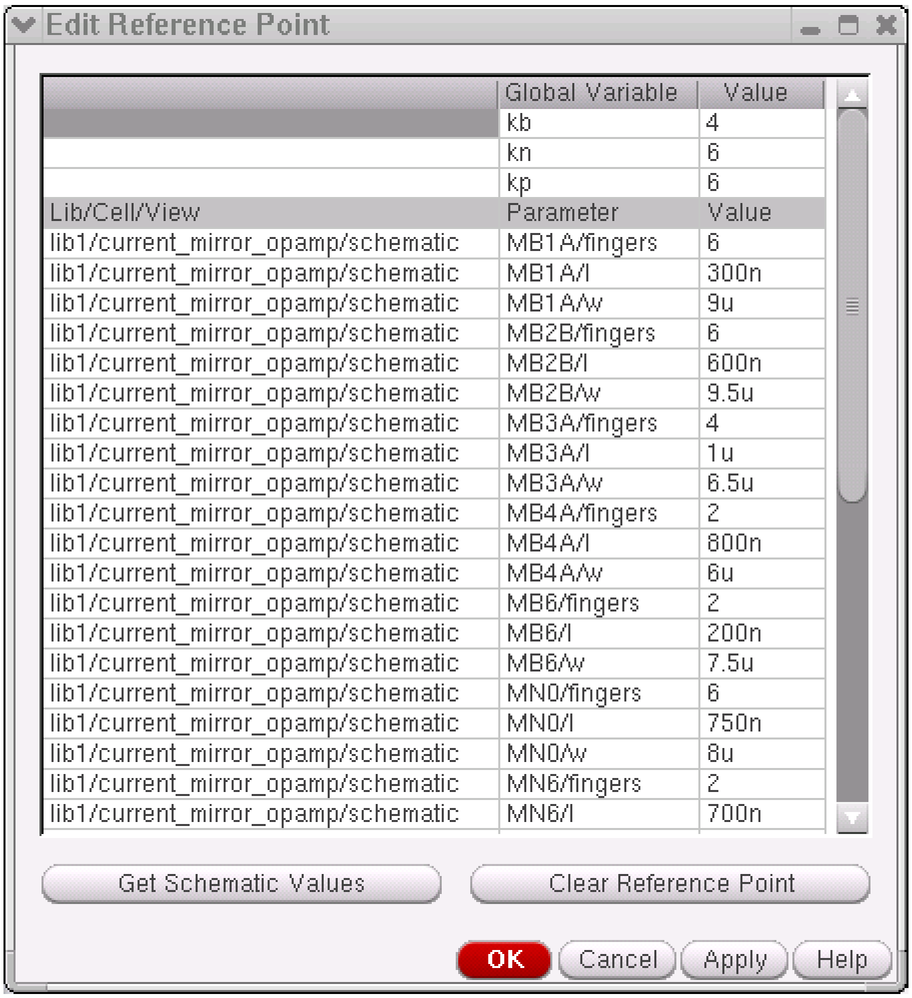
You can now utilize this point by selecting the *Use Reference Point as Starting Point* check box for Improve Yield or Global Optimization.

Viewing and Modifying the Current Reference Point

The current reference point is the last one created or modified, then saved to the setup database. You can view and modify this current reference point.

1. Click the *Edit Reference Point* icon in the Run Mode tool bar.

   The Edit Reference Point form appears, displaying the values for the current reference point.



	Global Variable	Value
	kb	4
	kn	6
	kp	6
Lib/Cell/View	Parameter	Value
lib1/current_mirror_opamp/schematic	MB1A/fingers	6
lib1/current_mirror_opamp/schematic	MB1A/l	300n
lib1/current_mirror_opamp/schematic	MB1A/w	9u
lib1/current_mirror_opamp/schematic	MB2B/fingers	6
lib1/current_mirror_opamp/schematic	MB2B/l	600n
lib1/current_mirror_opamp/schematic	MB2B/w	9.5u
lib1/current_mirror_opamp/schematic	MB3A/fingers	4
lib1/current_mirror_opamp/schematic	MB3A/l	1u
lib1/current_mirror_opamp/schematic	MB3A/w	6.5u
lib1/current_mirror_opamp/schematic	MB4A/fingers	2
lib1/current_mirror_opamp/schematic	MB4A/l	800n
lib1/current_mirror_opamp/schematic	MB4A/w	6u
lib1/current_mirror_opamp/schematic	MB6/fingers	2
lib1/current_mirror_opamp/schematic	MB6/l	200n
lib1/current_mirror_opamp/schematic	MB6/w	7.5u
lib1/current_mirror_opamp/schematic	MN0/fingers	6
lib1/current_mirror_opamp/schematic	MN0/l	750n
lib1/current_mirror_opamp/schematic	MN0/w	8u
lib1/current_mirror_opamp/schematic	MN6/fingers	2
lib1/current_mirror_opamp/schematic	MN6/l	700n

Important

If redundant parameters exist, they are shaded in yellow, and you will be prompted to delete them. Redundant parameters are parameters that are present in the starting point but are either deleted or disabled in the design space.

2. If desired, modify the values as described in step 2 of Creating, Viewing, and Modifying Reference Points:

3. Click *OK* to save your values and dismiss the form.

   If you are merely viewing the point, and haven't made any changes you want to save, click *Cancel*.

   Any reference point you create will remain the active reference point until you create or modify another point.

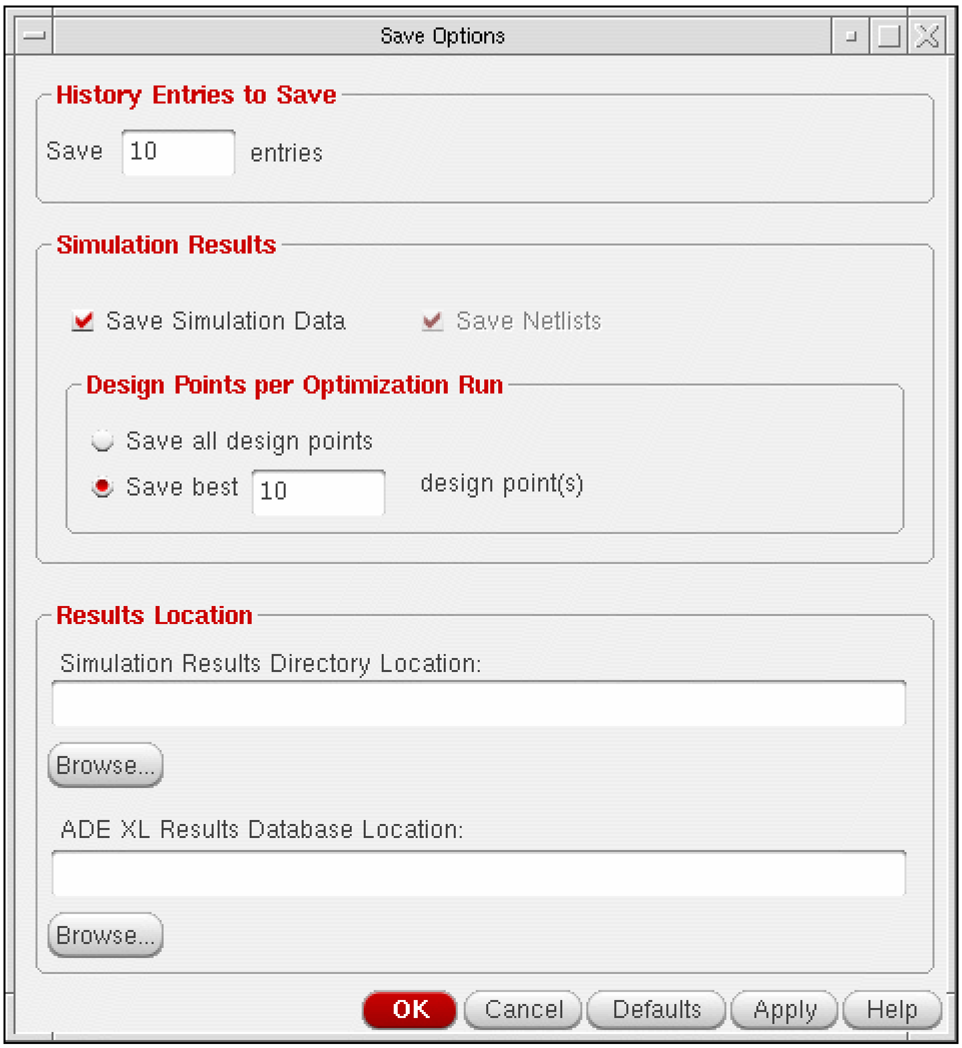
You can now utilize this point by selecting the *Use Reference Point as Starting Point* check box for Improve Yield or Global Optimization.

Specifying How Much Optimization Data to Save

To specify the number of design points for which you want to save simulation data, do the following:

1. In the ADE GXL session window, choose *ADE GXL – Options – Save*.

   The Save Options form appears.



2. In the *Design Points per Optimization Run* group box, select one of the following radio buttons:

Radio Button	**Description**
Save all design points	Saves data from all design points
Save best	Saves data from the specified number of best design points

3. (Optional) In the *Results DB* group box, either type the directory path where you want the program to write your setup and results database information in the *Results DB location* field, or do the following:

   a. Click *Browse*.

   b. On the form that appears, navigate to and select the directory where you want the program to write your setup and results database information.

   The program writes setup and results database information to `libraryName/cellName/adexl` in the specified directory.

   If you do not specify a setup and results database location, the program writes this information to `libraryName/cellName/adexl` for the current test. If your design library is set up as read-only, you can use this field to specify a writable location.

4. Click *OK*.

   The program applies the settings you specified.

See also

- Data Points—Definition on page 169
- Design Points—Definition on page 169
- Specifying Results Database Location” in the *Virtuoso Analog Design Environment XL User Guide*.

Data Points—Definition

A data point represents one simulation run with one set of parameter values and corners setup. For example, if you sweep *CAP* (a global variable) from `600p` to `800p` with a step value of `3`, you will have three data points. If you sweep more than one variable, each unique combination of values constitutes one data point. If you have two corners for temperature at `0` and `30` in addition to the *CAP* sweep through three values, you will have six data points:

Data Point	*CAP*	Corner Temperature
1	`600p`	`0`
2	`700p`	`0`
3	`800p`	`0`
4	`600p`	`30`
5	`700p`	`30`
6	`800p`	`30`

Design Points—Definition

A design point consists of the set of data points that represents one sweep value across all corners. For example, if you have two corners for temperature at `0` and `30`, and a sweep of *CAP* (a global variable) through three values (`600p`, `700p`, `800p`), each sweep across both corners constitutes a design point.

Design Point	*CAP*	Corner Temperatures
1	`600p`	`0`
	`600p`	`30`
2	`700p`	`0`
	`700p`	`30`
3	`800p`	`0`
	`800p`	`30`

5

High Yield Estimation

Large IC designs may contain 10k to10M replicated unit circuits, for example, SRAM cell and DFF. A rare failure event of a unit circuit may induce a not-so-rare failure of the entire system. The failure rate of a unit circuit is typically small (10-7 ~ 10-5). In this case, Brute-force Monte Carlo analysis may require millions or billions of simulations. However, High Yield Estimation provides a more efficient method when the simulation cost of Monte Carlo analysis is high. This chapter describes how to estimate the yield of high sigma designs using ADE GXL.

This chapter covers the following topics:

Performing High Yield Estimation

You can perform high yield estimation by using the following two methods:

- Worst Case Distance Method
- Scaled-Sigma Sampling Method

To open the high yield estimation form, do one of the following:

- Choose *Run – High Yield Estimation*.
- Select *High Yield Estimation* from the *Select a Run Mode* drop-down list on the Run toolbar, and then click the *Simulation Options* button on the Run toolbar.

  The High Yield Estimation form appears. You can use this form to perform the high yield estimation based on the Scaled-Sigma Sampling and Worst Case Distance methods. Refer to the next sections for more details.

Worst Case Distance Method

In the Worst Case Distance method, the high yield estimation finds the shortest distance—referred to as Worst Case Distance or WCD—from the nominal point to the specification boundary in the process/mismatch parameter space. The worst case distance is a good indicator of circuit yield, where yield in percentage is approximately equal to

$$\frac{1}{\sqrt{2}\pi}\int_{-\infty}^{wcd} e^{-t^2/2}\,dt = \frac{1}{2}\left[1 + erf\left(\frac{wcd}{\sqrt{2}}\right)\right]$$

where, `erf` is the error function.

Note: WCD provides accurate yield estimation when the specification boundary is linear in the process or mismatch parameter space. Strong non-linearity of the specification boundary can cause difficulty in finding the WCD points. A non-linear specification however may not result in a non-linear specification boundary in statistical space.

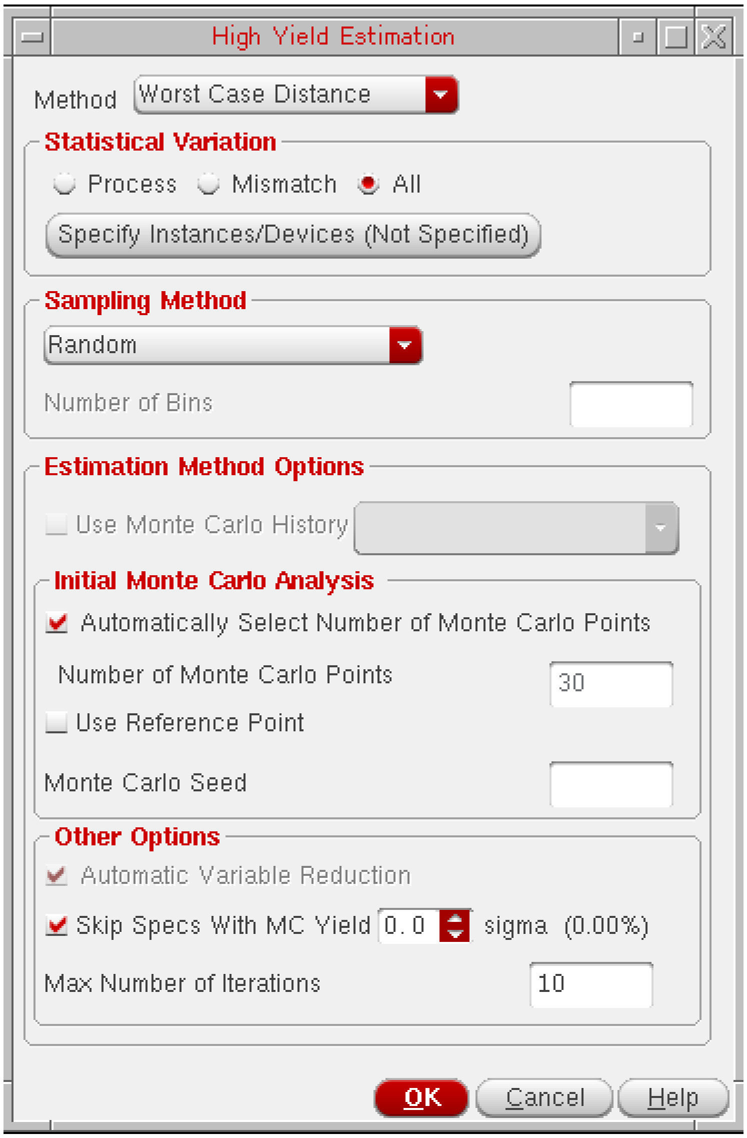
The Worst Case Corners method supports only statistical parameters that follow a normal distribution. It begins with a Monte Carlo Sampling run, uses the Monte Carlo results to filter non-high yield specifications (for which the Monte Carlo run gives accurate yield estimates), and then applies the WCD method on each high yield specification by doing the following:

- Reads the process and mismatch parameter information from the Monte Carlo results
- Performs parameter reduction based on the Monte Carlo results
- Runs multiple sensitivity analysis iterations to find the WCD

You can improve the circuit yield by creating a statistical corner from the worst case distance point. For more details, refer to Creating Statistical Corners From a Worst Case Distance Analysis on page 188.

Before you perform high yield estimation, ensure that the Range and tolerance (`tol`) type specifications are disabled or deleted in the Outputs Setup tab. This is because these two specifications are not supported for Worst Case Distance. You can also convert the range and tolerance (`tol`) type specifications into two separate specifications for the min and max boundaries. For more information about specifications, see General Specifications.

To perform the high yield estimation by using worst case distance, select *Method* as `Worst Case Distance` in the High Yield Estimation form. The following fields are displayed in the High Yield Estimation form as shown below:



1. While running High Yield Estimation, you can choose to vary process statistical variables, mismatch statistical variables, or both. If you run only one type of statistical variable, the other variables are set to fixed values.

In the *Statistical Variation* group box, select one of the following statistical variations:

Process	for process statistical variations
Mismatch	for per-instance statistical variations
All	for both process and per-instance statistical variations

Important

You must define your models so that they respond to the statistical variations you choose. You must specify the file containing your models on the Model Library Setup form. For a Spectre circuit simulator example of how to define your models, see "Specifying Parameter Distributions Using Statistics Blocks" in *Virtuoso Spectre Circuit Simulator User Guide*.

2. By default, mismatch variations are applied to all subcircuit instances in the design. Click the *Specify Instances/Devices* button to specify the sensitive instances, devices, and parameters that you want to either include or exclude for applying mismatch variations.

   The *Select Instances and Parameters for Mismatch* form is displayed. For more information about how to use this form, refer to Selecting Instances and Parameters for Mismatch Variation.

3. From the *Sampling Method* list, select a statistical sampling method from the drop-down list. You can select any one of the following sampling methods:

   - *Random*
   - *Latin Hypercube*
   - *Low Discrepancy Sequence*

   For more details about these methods, see Sampling Methods in *Analog Design Environment XL User Guide*.

4. If the selected sampling method is *Latin Hypercube*, specify the number of bins (subdivisions) for the *Latin Hypercube* method in the *Number of Bins* field. For more information, see Number of Bins in *Analog Design Environment XL User Guide*.

5. If you have already run a Monte Carlo simulation, you can use the process and mismatch data from the history of that run. To do that, select the *Use Monte Carlo History* option in the *Estimation Method Options* section and select a reference Monte Carlo run history from the list of available histories given in the drop-down list next to this option.

It is essential that the simulation data of the selected history contains the process and mismatch data. If any one of these data is not available, the following error message is displayed.

In this case, you can clear the *Use Monte Carlo History* option and run a fresh Monte Carlo simulation or choose another history that contains the process and mismatch data.

Note: When you choose to use an existing Monte Carlo history as reference, you can only specify options in the *Other Options* section. Values for the remaining options are taken from the referenced Monte Carlo history.

6. The *Automatically Select Number of Monte Carlo Points* check box in the *Initial Monte Carlo Analysis* section is selected by default. This is used to enable the automatic selection of number of Monte Carlo points. When you select this check box, the *Number of Monte Carlo Points* and *Automatic Variable Reduction* fields become unavailable. To manually provide the number of Monte Carlo points to be simulated, disable this check box.

7. In the *Number of Monte Carlo Points* field, type the number of Monte Carlo points you want to simulate.

8. Select the *Use Reference Point* check box if you have created a reference point and want to use that point as the starting point for the run.

9. (Optional) If you want to specify a different seed for the Monte Carlo analysis, enter the seed number in the *Monte Carlo Seed* field. By specifying the same seed, you can reproduce a previous experiment. If you do not specify a seed, the value `12345` is used.

10. By default, the *Automatic Variable Reduction* check box is disabled. To enable this check box, deselect the *Automatically Select Number of Monte Carlo Points* check box provided in the *Sampling Method* section. The *Automatic Variable Reduction* option reduces the set of statistical variables by eliminating insignificant variables (variables that have no variation or have no influence on the WCD point). Insignificant variables bring noise and require more simulations for sensitivity analysis. Therefore, it is recommended to enable variable reduction.

11. To ignore the specifications for which Monte Carlo yield is less than a specified percentage, select the *Skip the Specs Whose MC Yield <* checkbox and specify a sigma value.

The default value of this field is `3` sigma; therefore, specifications for which the Monte Carlo yield is less than 99.73% are ignored.

12. If you want to run high yield estimation on all the specifications, disable the *Skip the Specs Whose MC Yield <* check box.

13. In the *Max Number of Iterations* field, specify the maximum number of iterations to be run for each specification.

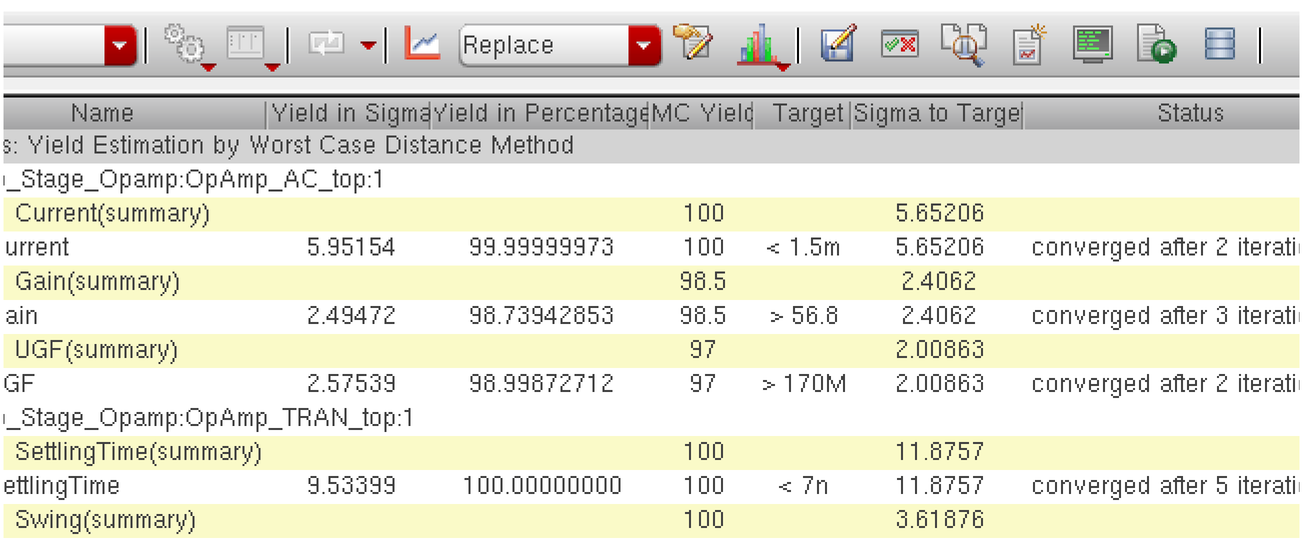
The default number of iterations is `10`.

Running High Yield Estimation Using WCD

Click the *Run Simulation* button on the Run toolbar to start the high yield estimation run.

The *Run Log / Log File Viewer* form appears displaying information regarding the progress of the initial Monte Carlo Sampling run, the yield estimate at each iteration, and the summary of the High Yield Estimation run. The log file also displays the sigma of the statistical variable, which helps easy understand the results when you work with different units. The log file also displays the statistical parameter contribution values for the WCD point and device contribution values for the squared WCD values.

The results for the run are displayed in the Results tab, as shown below.



Name	Yield in Sigma	Yield in Percentage	MC Yield	Target	Sigma to Targe	Status
s: Yield Estimation by Worst Case Distance Method						
ι_Stage_Opamp:OpAmp_AC_top:1						
Current(summary)			100		5.65206	
urrent	5.95154	99.99999973	100	< 1.5m	5.65206	converged after 2 iterati
Gain(summary)			98.5		2.4062	
ain	2.49472	98.73942853	98.5	> 56.8	2.4062	converged after 3 iterati
UGF(summary)			97		2.00863	
GF	2.57539	98.99872712	97	> 170M	2.00863	converged after 2 iterati
ι_Stage_Opamp:OpAmp_TRAN_top:1						
SettlingTime(summary)			100		11.8757	
ettlingTime	9.53399	100.00000000	100	< 7n	11.8757	converged after 5 iterati
Swing(summary)			100		3.61876	
wing	4.09053	99.99569621	100	> 1.38	3.61876	converged after 2 iterati

The Results tab shows the following columns:

- *Name*: Displays the name of specification.
- *Yield in Sigma*: Displays the yield value in sigma. This value is calculated as shown below:

  ```
  Yield in Sigma = sqrt(2) * erfinv (Yield in Percentage/100)
  ```

  where, `erfinv` is the inverse error function.

  If the yield in sigma is greater than `8.2`, the yield in percentage is displayed as `100%`.

- *Yield in Percentage*: Displays the yield value in percentage. This value is calculated as shown below:

  ```
  Yield in Percentage = 0.5 + 0.5 * erf (WCD/sqrt(2))
  ```

  where, `erf` is the error function.

  Note: The yield in percentage value is by default displayed with 10 digits. To change the number of digits to be displayed for this value, set the value of variable *digitsToShowForYieldInPercentage* environment variable. You can display a maximum of 53 digits for these values.

- *MC Yield*: Displays the yield value from the Monte Carlo run.
- *Target*: Displays the target to be achieved for the given specification.
- *Sigma to Target*: Displays the sigma to target value.
- *Status*: Displays the convergence status for each specification.

  The tool uses the following two convergence criteria:

  - The predicted WCD point is on the specification boundary (within tolerance < 0.02). The log file reports the 'Spec value error' at each iteration.

    ```
    spec_value_error_ratio = abs(spec_value -
    spec_target)/max(abs(nominal_spec_value - spec_target), 6*spec_sigma)
    ```

  - The angle between the gradient vector and the statistical variable vector is < 8 deg. The log file reports the 'Gradient direction error' at each iteration.

  The convergence status can be any of the following:

Status	Description
converged after `x` *iterations*	The yield estimate converged after *x* iterations. For example, the yield estimate for the `Slew_rate` specification converged after `2` iterations.

Status	Description
skipped because the Monte Carlo yield is less than <value in percent>	Yield estimation was skipped for the specification because the Monte Carlo yield estimate was too low. The low yield threshold value is specified in the run option form in the *Skip specs with MC Yield < value* field, which is by default set to `3 sigma` or `99.73%`.
least error WCD, did not converge after x iterations	Yield estimation did not converge after the maximum number of iterations has completed. The specification failure boundary is strongly non-linear or the maximum number of iterations is too small. The yield estimate with the least error among iterations is reported. For details on the High Yield Estimation criteria, see convergence criteria.
lower boundary, did not converge after x iterations	Yield estimation did not converge because the specification has an unrealistic yield estimate which is larger than 12 sigma in yield after the maximum number of iterations. The yield estimate increased at each iteration, but never converged. For details on the High Yield Estimation criteria, see convergence criteria.
estimate based on MC data / lower boundary / least error WCD, stopped because evaluating of the WCD point sensitivity failed on iteration x	Yield estimation stopped before reaching the maximum number of iterations because of a simulation or measurement error in evaluating the WCD point sensitivity. The lower boundary is reported if it is identified, if not, the yield estimate with the least error among iterations is reported.
estimate based on MC data / lower boundary / least error WCD, stopped because evaluation of the WCD point failed on iteration x	Yield estimation stopped before reaching the maximum number of iterations because of a simulation or measurement error in evaluating the WCD point. The lower boundary is reported if it is identified, if not, the yield estimate with the least error among iterations is reported. If the run was stopped on the first iteration, the estimate based on the Monte Carlo result is reported.

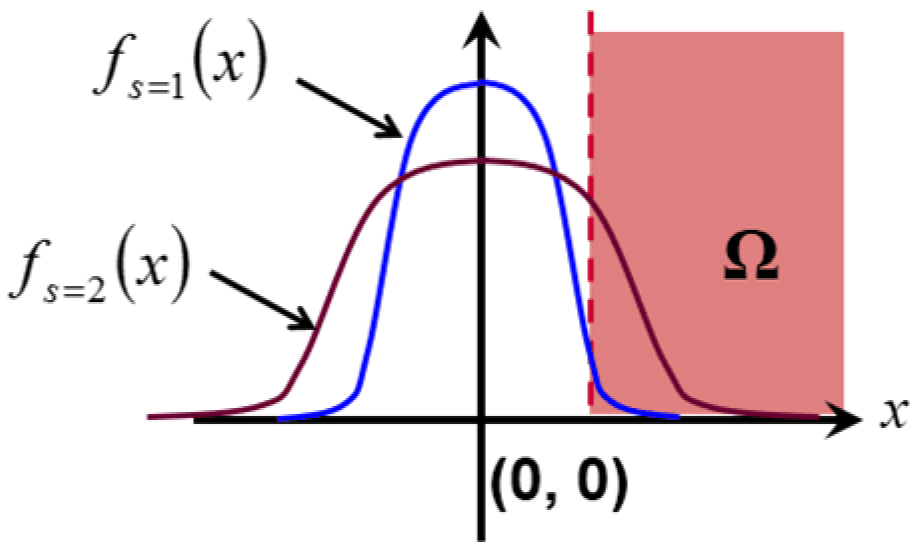
Scaled-Sigma Sampling Method

Scaled-sigma sampling method generates samples where the standard deviation has been scaled up. As a result, a larger number of samples fall into the failure region of the distorted distribution. The failure rate is then estimated from the scaled samples.

Following are the advantages of Scaled-Sigma Sampling method:

- Efficient for high dimensionality (very large numbers of devices and statistical parameters)
- Accurate even for cases of high non-linearity
- Efficient when the design is constrained by a large number of specifications

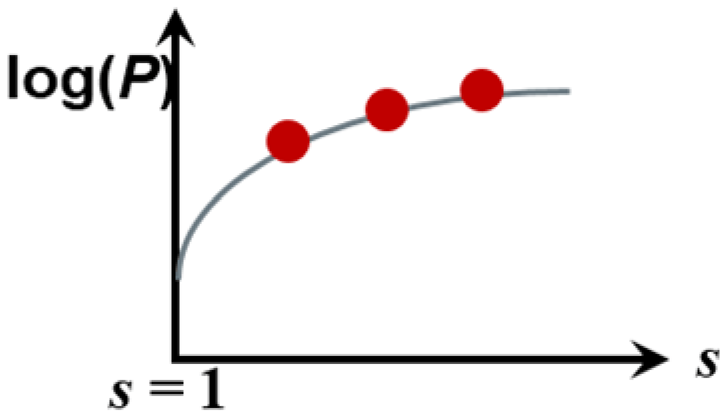
This figure shows the unscaled performance distribution (scaling factor s=1) compared to the scaling factor s=2.



Here, x = process parameter and Ω = failure region.

The failure rate can be calculated as[1]:

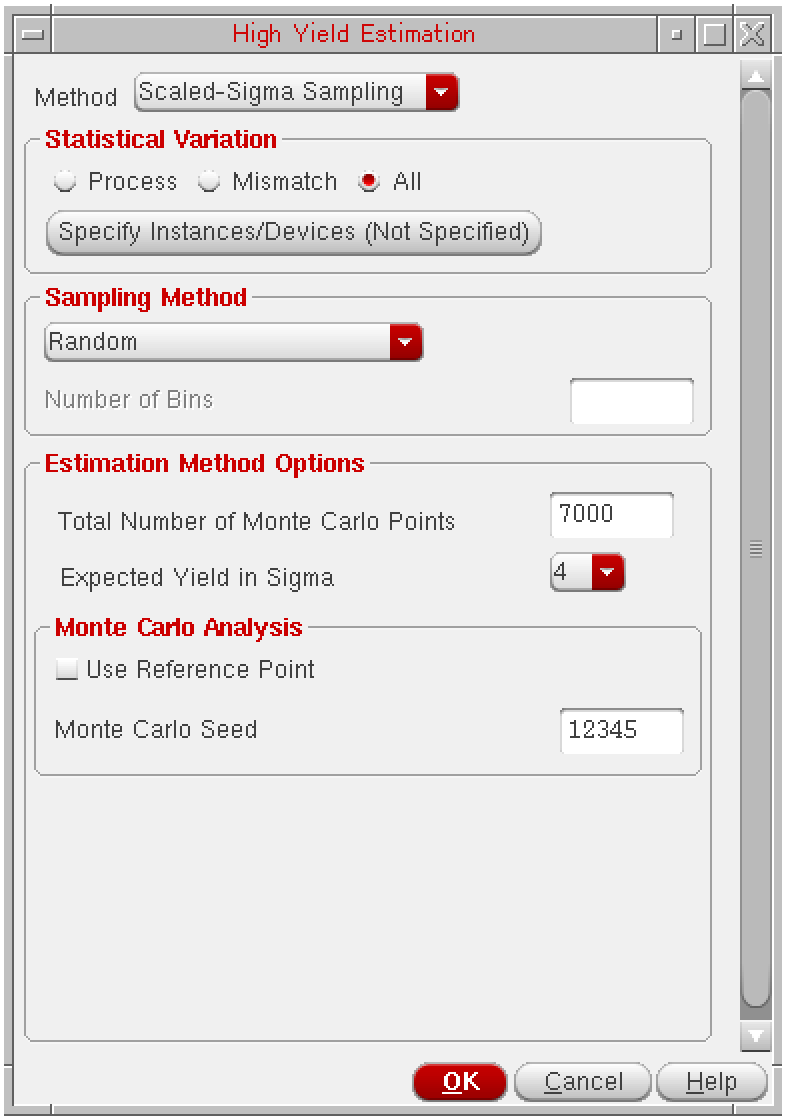
$$\log(P) \approx \alpha + \beta\log(s) + \gamma \;\; s^2$$



The above equation models the failure rate as a function of the scaling factor. The model has very few constraints on the failure region and can target multiple failure regions. The model is constructed based on a set of scaled Monte Carlo runs. Then, the unscaled yield estimate (s=1) can be found.

1. Shupeng Sun, Xin Li, Hongzhou Liu, Kangsheng Luo, and Ben Gu, "Fast statistical analysis of rare circuit failure events via scaled-sigma sampling for high-dimensional variation space," IEEE/ACM International Conference on Computer-Aided Design (ICCAD), pp. 478-485, 2013.

To perform high yield estimation by using scaled-sigma sampling, select *Method* as `Scaled-Sigma Sampling` in the High Yield Estimation form, which is shown below:



For more information about the *Statistical Variation* and *Sampling Method* sections shown in the form above, see Worst Case Distance Method on page 173.

1. In the *Estimation Method Options* section,

   a. In the *Total Number of Monte Carlo Points* field, specify the total number of Monte Carlo points that will be simulated over the set of scaled sigma runs. The default value is `7000`.

   Note: The Scaled-Sigma Sampling method has multiple Monte Carlo runs with different scaling factors. Therefore, the *Total Number of Monte Carlo Points* are used instead of *Number of Monte Carlo points* for a Monte Carlo run as used in the Worst Case Distance method.

b. In the *Expected Yield in Sigma* drop-down list, specify the yield value that closely matches your yield verification requirement. The available options are `4`, `5`, and `6`.

2. In the Monte Carlo Analysis section, select the *Use Reference Point* check box if you have created a reference point and want to use this point as the starting point for the run.

Note: A reference point must be created if swept values are specified for enabled global variables or parameters.

3. (Optional) If you want to specify a different seed for the Monte Carlo analysis, enter the required seed value in the *Monte Carlo Seed* field. When you specify the seed that is used in the Monte Carlo run, you can reproduce the specific experiment. If you do not specify a seed, the value `12345` is used.

Running High Yield Estimation Using Scaled-Sigma Sampling

Click the *Run Simulation* button on the *Run* toolbar to start the high yield estimation run.

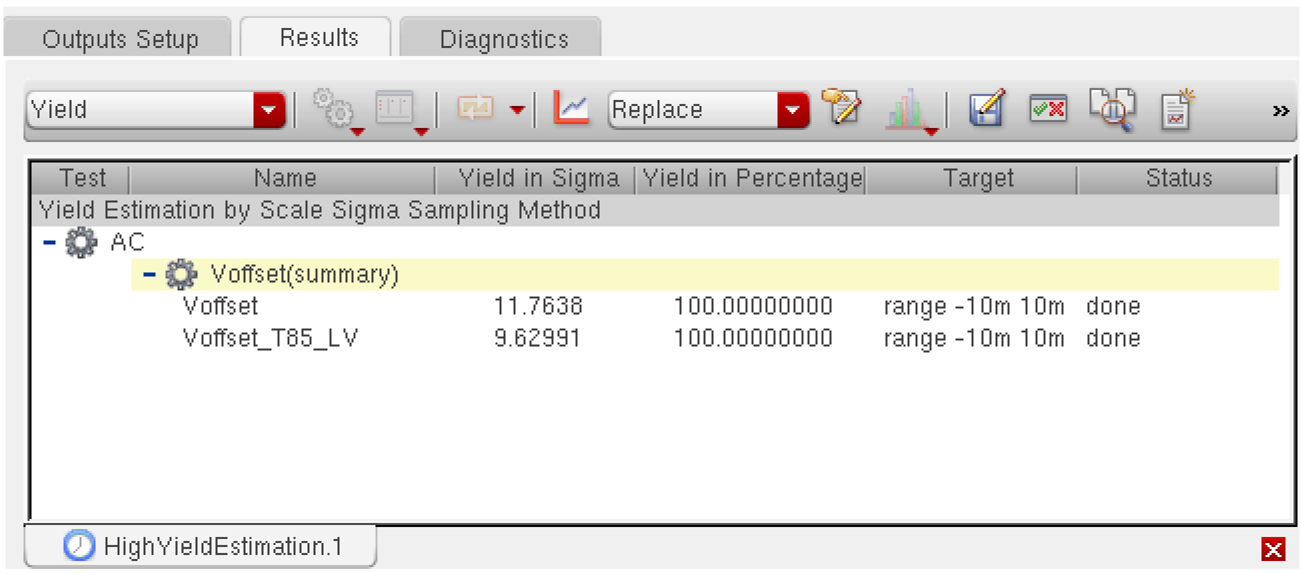
The *Run Log/Log File Viewer* window appears displaying information about the progress of the Monte Carlo Sampling run, the scaled sigma value, the Monte Carlo yield and the predicted yield at each iteration, and the summary of the High Yield Estimation run.

```
Monte Carlo Sampling started.

Monte Carlo Sampling

Scaled Sigma: 6.08466
Monte Carlo Yield:
        Test: AC Corner: Nominal Spec: AC.Voffset Yield: 88.5%
        Test: AC Corner: T85_LV Spec: AC.Voffset Yield: 85.2%
Monte Carlo Sampling #6 completed.
Current time: Tue Oct  7 17:05:23 2014


Predicted Yield:
        Test: AC Corner: Nominal Spec: AC.Voffset
        Model: log(failure rate) = 1.73449 + -0.99763 * log(scale) + -74.3194 / scale^2
        Yield in Sigma: 11.7638, 90% Confidence interval: (10.2842, 13.8508)
        Functional Yield in Sigma: > 17.5126, the yield estimate is the lower boundary because the yield is too high.

        Test: AC Corner: T85_LV Spec: AC.Voffset
        Model: log(failure rate) = -1.58623 + 0.600468 * log(scale) + -47.9756 / scale^2
        Yield in Sigma: 9.62991, 90% Confidence interval: (7.7065, 11.2742)
        Functional Yield in Sigma: > 17.5126, the yield estimate is the lower boundary because the yield is too high.
```

The run log reports the 90% confidence interval of the yield estimate in parenthesis, for example, (`10.2842, 13.8508`) shown in the figure above.

The runlog file also displays the functional yield in sigma value, which is an estimate based on a functional pass or fail criteria. A failed point is counted only when the simulation or measurement evaluation fails.

The results for the run are displayed on the *Results* tab, as shown below.



The *Results* tab includes the following fields:

- *Test*—Displays the name of the test.
- *Name*—Displays the name of the specification.
- *Yield in Sigma*—Displays the yield value. This value is calculated as shown below:

  `Yield in Sigma = sqrt(2) * erfinv (Yield in Percentage/100)`

  where, `erfinv` is the inverse error function.

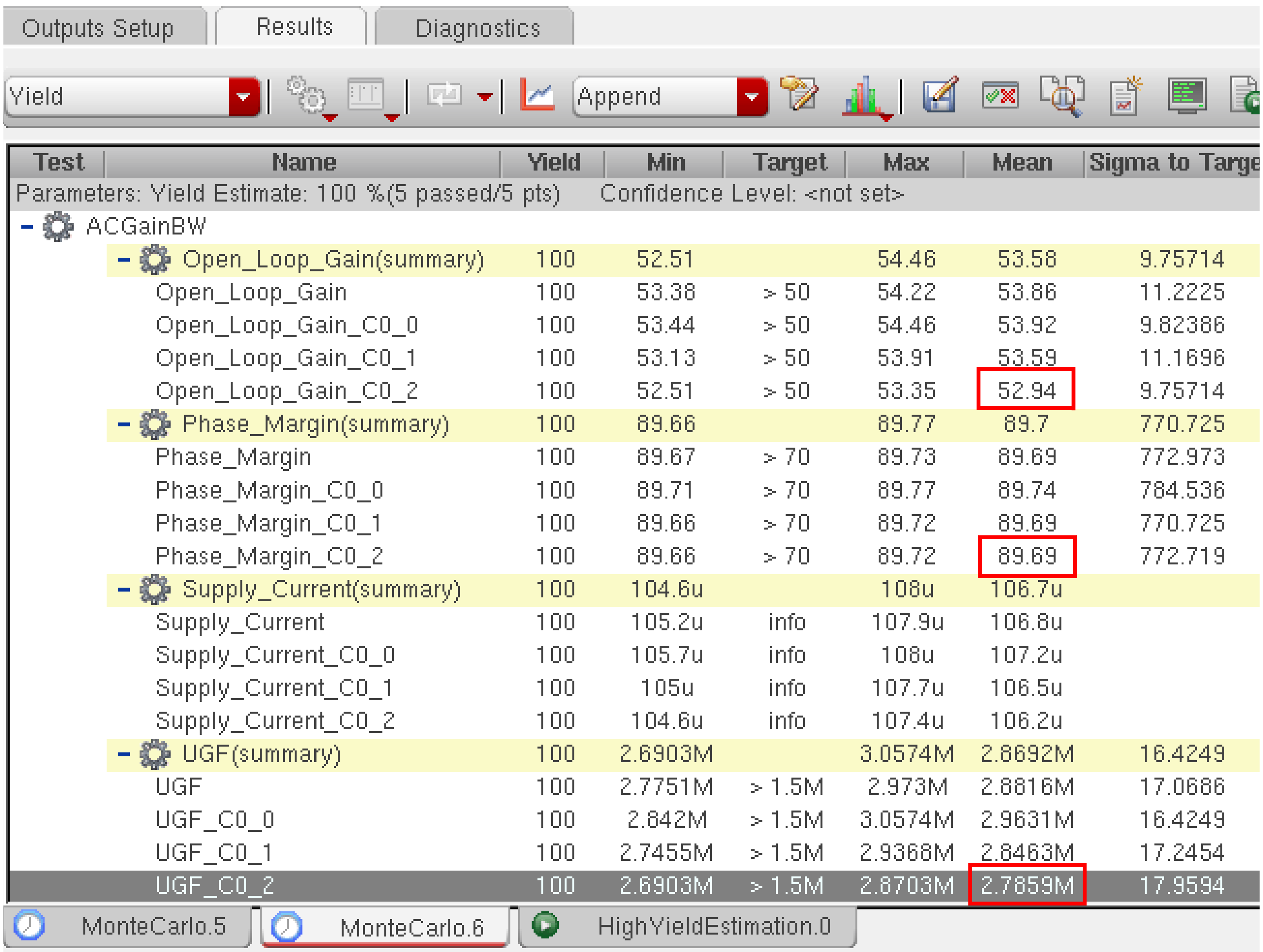
  If the yield in sigma is greater than `8.2`, the yield in percentage is displayed as `100%`.
- *Yield in Percentage*—Displays the yield value in percentage.
  The yield in percentage value is by default displayed with 10 digits. To change the number of digits to be displayed, set the value in the *`digitsToShowForYieldInPercentage`* environment variable. You can display a maximum of 53 digits.
- *Target*—Displays the target to be achieved for the given specification.

- *Status*—Displays the convergence status for the given specification.

High Yield Estimation with Multiple Corners

If you enable multiple corners for simulation, Monte Carlo simulation is performed for all of the enabled corners. However, for high yield estimation, only a single corner is selected for each specification. The selected corner is the one that has the worst mean value out of all the corners in the Monte Carlo results for a given specification.

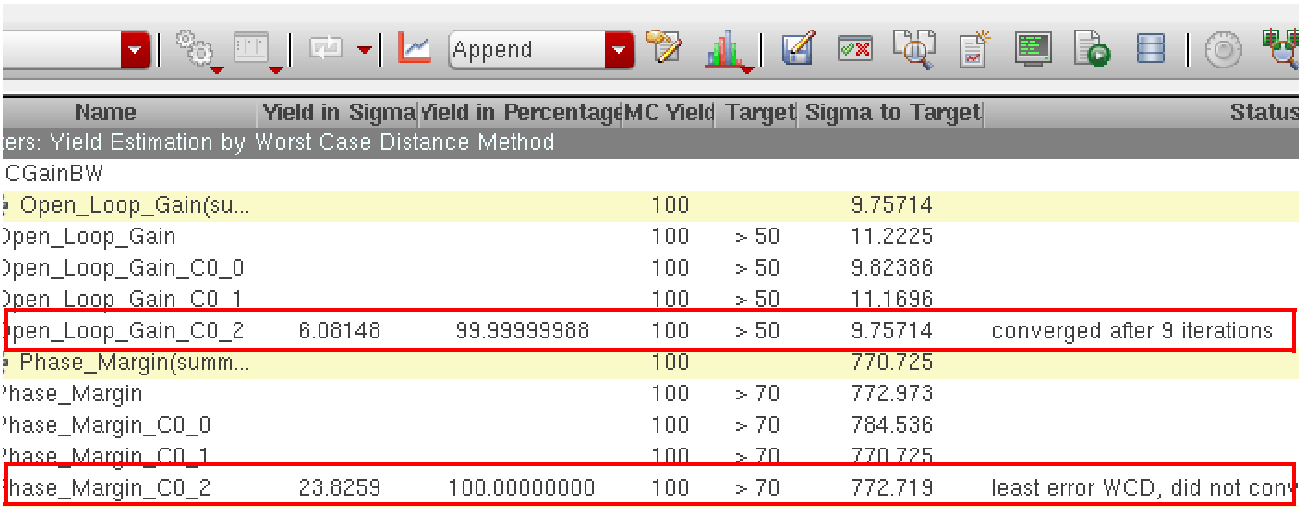
Consider an example of the Monte Carlo results for a test with multiple corners, as shown below.



Test	Name	Yield	Min	Target	Max	Mean	Sigma to Targe
Parameters: Yield Estimate: 100 %(5 passed/5 pts)	Confidence Level: <not set>						
ACGainBW							
	Open_Loop_Gain(summary)	100	52.51		54.46	53.58	9.75714
	Open_Loop_Gain	100	53.38	> 50	54.22	53.86	11.2225
	Open_Loop_Gain_C0_0	100	53.44	> 50	54.46	53.92	9.82386
	Open_Loop_Gain_C0_1	100	53.13	> 50	53.91	53.59	11.1696
	Open_Loop_Gain_C0_2	100	52.51	> 50	53.35	52.94	9.75714
	Phase_Margin(summary)	100	89.66		89.77	89.7	770.725
	Phase_Margin	100	89.67	> 70	89.73	89.69	772.973
	Phase_Margin_C0_0	100	89.71	> 70	89.77	89.74	784.536
	Phase_Margin_C0_1	100	89.66	> 70	89.72	89.69	770.725
	Phase_Margin_C0_2	100	89.66	> 70	89.72	89.69	772.719
	Supply_Current(summary)	100	104.6u		108u	106.7u	
	Supply_Current	100	105.2u	info	107.9u	106.8u	
	Supply_Current_C0_0	100	105.7u	info	108u	107.2u	
	Supply_Current_C0_1	100	105u	info	107.7u	106.5u	
	Supply_Current_C0_2	100	104.6u	info	107.4u	106.2u	
	UGF(summary)	100	2.6903M		3.0574M	2.8692M	16.4249
	UGF	100	2.7751M	> 1.5M	2.973M	2.8816M	17.0686
	UGF_C0_0	100	2.842M	> 1.5M	3.0574M	2.9631M	16.4249
	UGF_C0_1	100	2.7455M	> 1.5M	2.9368M	2.8463M	17.2454
	UGF_C0_2	100	2.6903M	> 1.5M	2.8703M	2.7859M	17.9594

In this case, the worst mean value for each specification is for corner `C0_2`. As a result, for each specification, the tool performs high yield estimation by using corner `C0_2`.

After the worst case distance analysis is complete, the yield in sigma, yield in percentage, and convergence status are shown only for those corners that were used to perform the high yield estimation, as shown below.

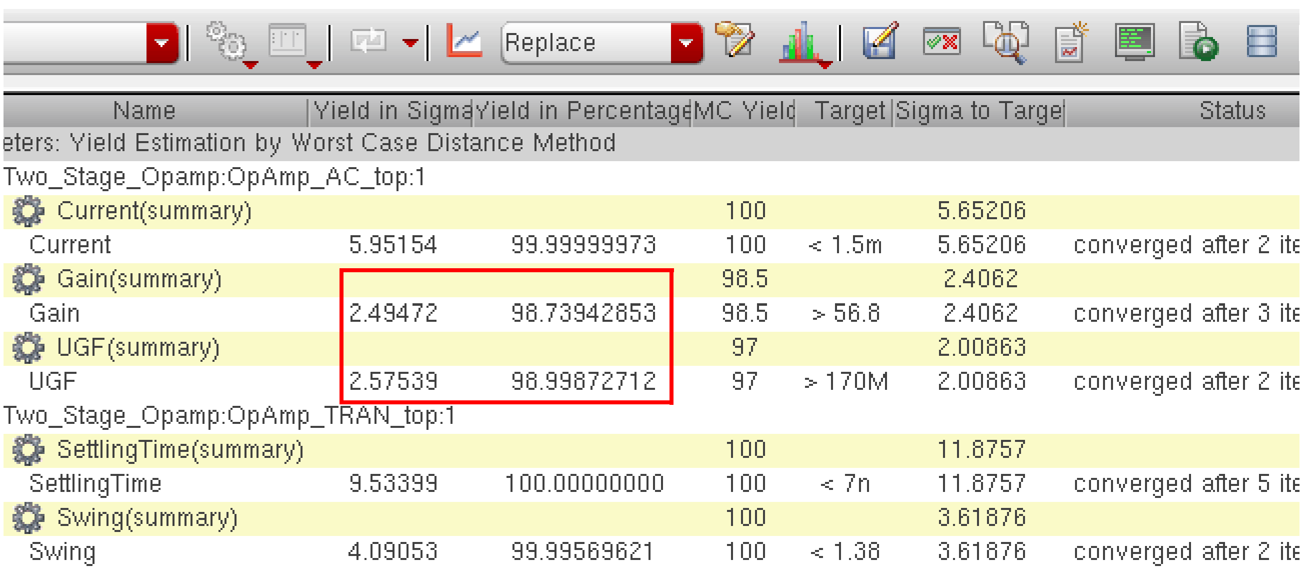


Name	Yield in Sigma	Yield in Percentage	MC Yield	Target	Sigma to Target	Status
ers: Yield Estimation by Worst Case Distance Method						
CGainBW						
Open_Loop_Gain(su...			100		9.75714	
)pen_Loop_Gain			100	> 50	11.2225	
)pen_Loop_Gain_C0_0			100	> 50	9.82386	
)pen_Loop_Gain_C0_1			100	> 50	11.1696	
)pen_Loop_Gain_C0_2	6.08148	99.99999988	100	> 50	9.75714	converged after 9 iterations
Phase_Margin(summ...			100		770.725	
'hase_Margin			100	> 70	772.973	
'hase_Margin_C0_0			100	> 70	784.536	
'hase_Margin_C0_1			100	> 70	770.725	
'hase_Margin_C0_2	23.8259	100.00000000	100	> 70	772.719	least error WCD, did not con

Creating Statistical Corners From a Worst Case Distance Analysis

After running the high yield estimation run, if you find that the required yield value is not achieved for any specification, you can create statistical corner based on the worst case distance point for that specification. You can use these new corners to run optimization or manually tune the design accordingly.

For example, if you run high yield estimation for a design without skipping the specs that have monte carlo yield of less than 3 sigma, the results might show specifications that have yield value less that 3 sigma, as shown below:

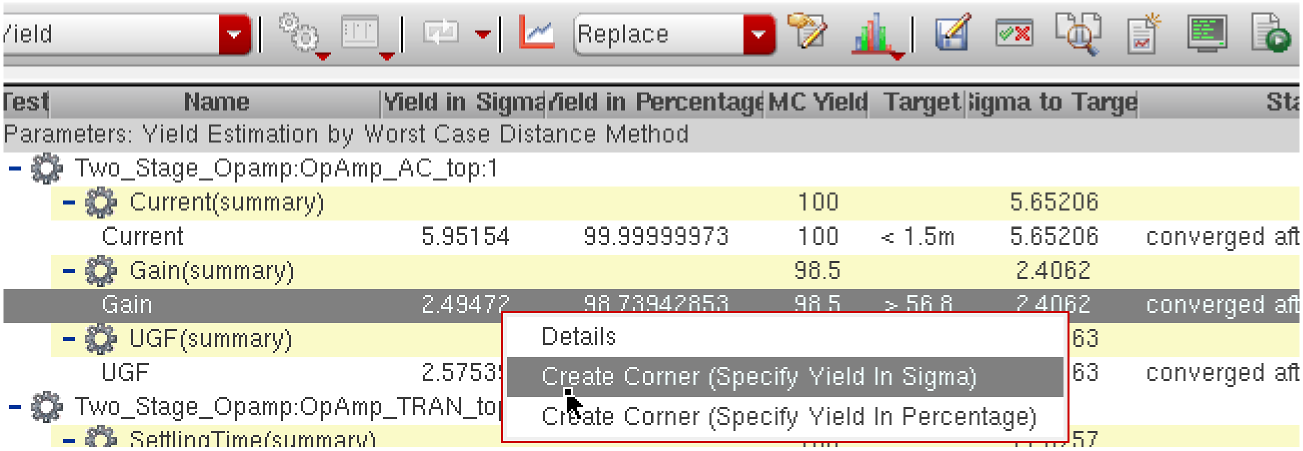


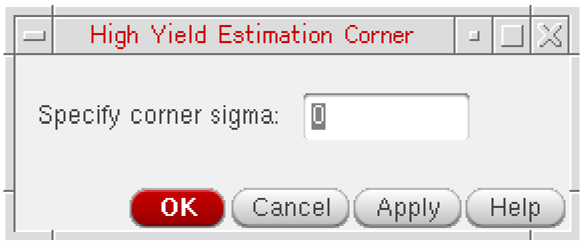
Name	Yield in Sigma	Yield in Percentage	MC Yield	Target	Sigma to Targe	Status
eters: Yield Estimation by Worst Case Distance Method						
Two_Stage_Opamp:OpAmp_AC_top:1						
Current(summary)			100		5.65206	
Current	5.95154	99.99999973	100	< 1.5m	5.65206	converged after 2 ite
Gain(summary)			98.5		2.4062	
Gain	2.49472	98.73942853	98.5	> 56.8	2.4062	converged after 3 ite
UGF(summary)			97		2.00863	
UGF	2.57539	98.99872712	97	> 170M	2.00863	converged after 2 ite
Two_Stage_Opamp:OpAmp_TRAN_top:1						
SettlingTime(summary)			100		11.8757	
SettlingTime	9.53399	100.00000000	100	< 7n	11.8757	converged after 5 ite
Swing(summary)			100		3.61876	
Swing	4.09053	99.99569621	100	< 1.38	3.61876	converged after 2 ite

Note that in the above results, the monte carlo yield for Gain and UGF is less than 3 sigma. To improve the performance for these specifications, you can create a statistical corner based on the Worst Case Distance (WCD) point of these specifications. For this, you can specify a target yield value greater than three sigma, either in terms of sigma or in terms of percentage value.

To create a WCD corner based on yield in sigma, right-click on a specification and choose *Create Corner (Specify Yield In Sigma)*.

The High Yield Estimation Corner form appears:





In the High Yield Estimation Corner form, do the following:

1. In the *Specify corner sigma* field, specify a target sigma value to which you want to extend the WCD point.
2. Click *OK*.

   A statistical corner is created by using the specification values that can improve the circuit yield to the target sigma value. The name of the statistical corner is prefixed with `WCD_`.

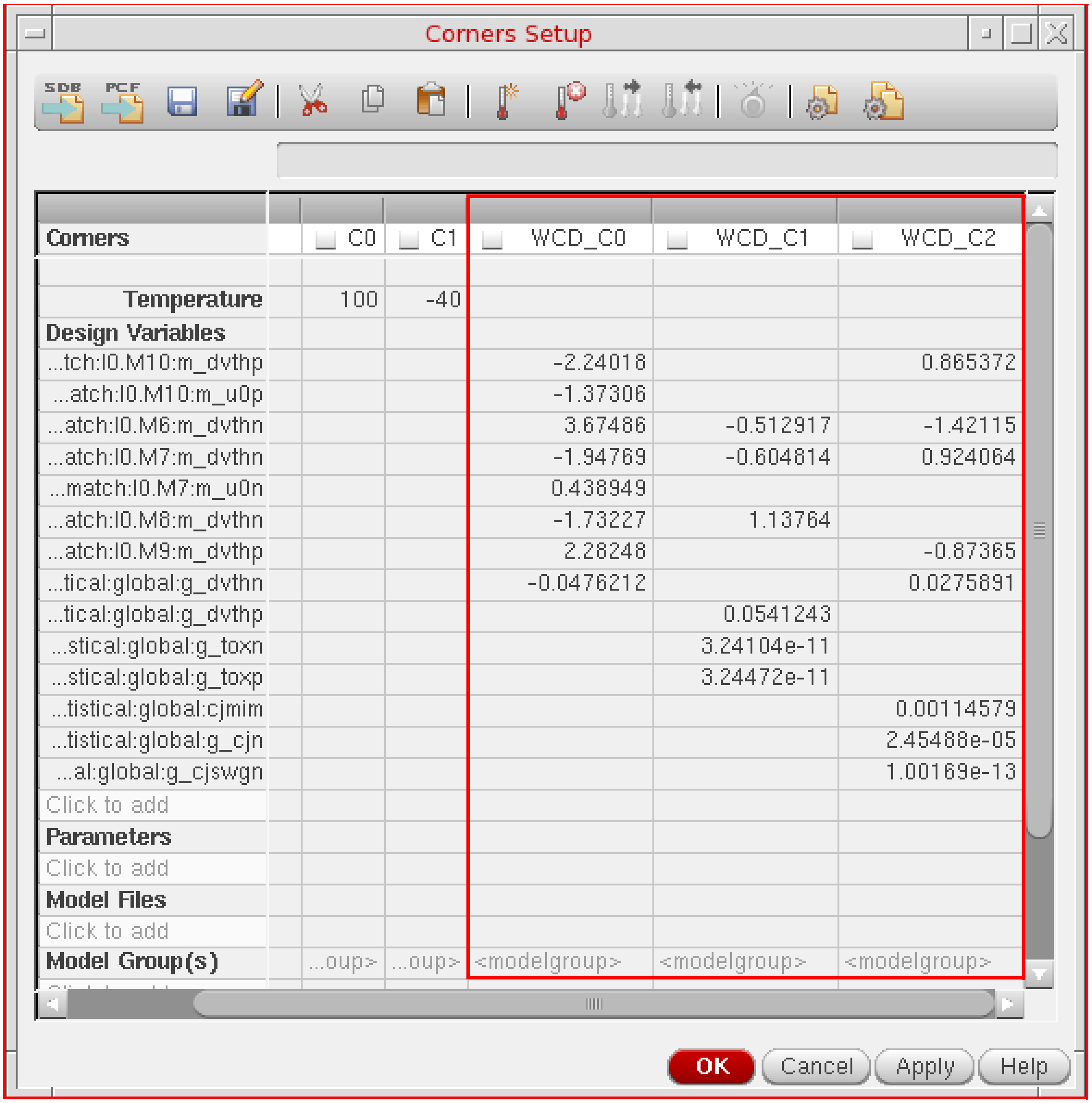
   The new corner is added to the Data View and the **Corners Setup** form.

Alternatively, you can choose *Create Corner (Specify Yield In Percentage*) and specify the target yield value to be achieved in terms of percentage.

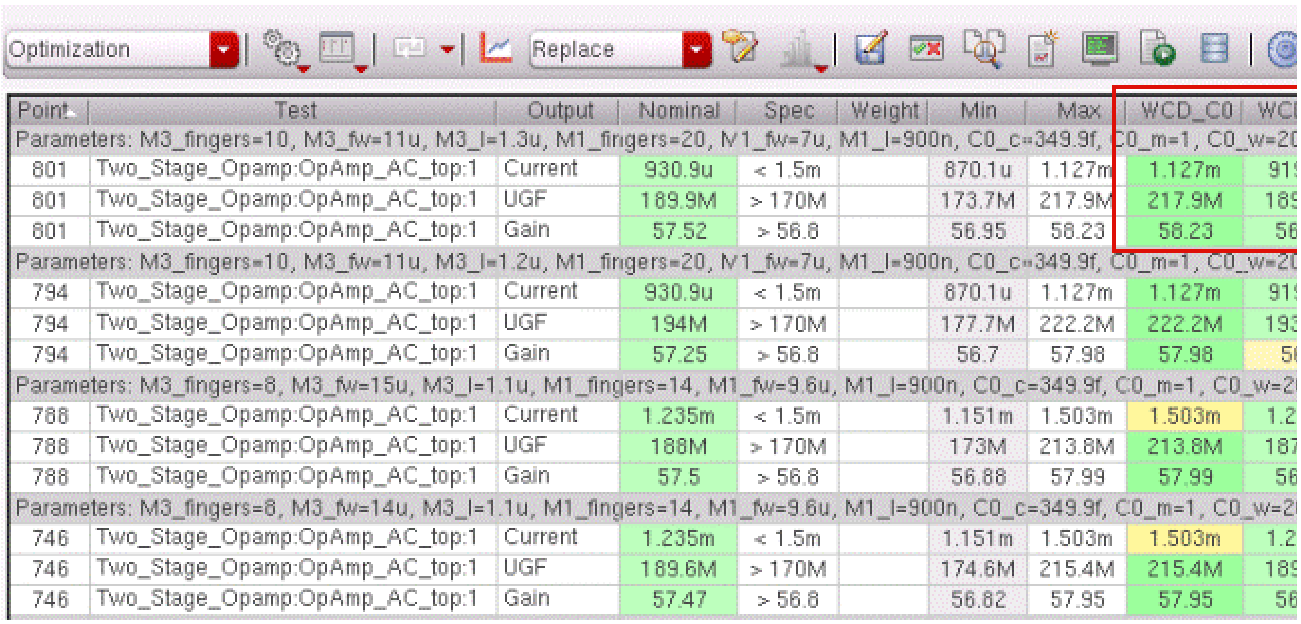
Tip

For a normal design, you can set for a corner sigma value equal to 3.0. However, for a high yield design, you can set a corner sigma value equal to 6.0. A good practice is to increase the sigma value by 0.5 to 1.0 in each iteration instead of increasing it by large amount.

In the example shown above, let us create three statistical corners by setting the corner sigma for Current, Gain, and UGF equal to `6`, `3`, and `3`, respectively. In this case, the corners are created as shown below:



Now, if you enable corners and run optimization on the design, the results for the new statistical corners appear as shown below:

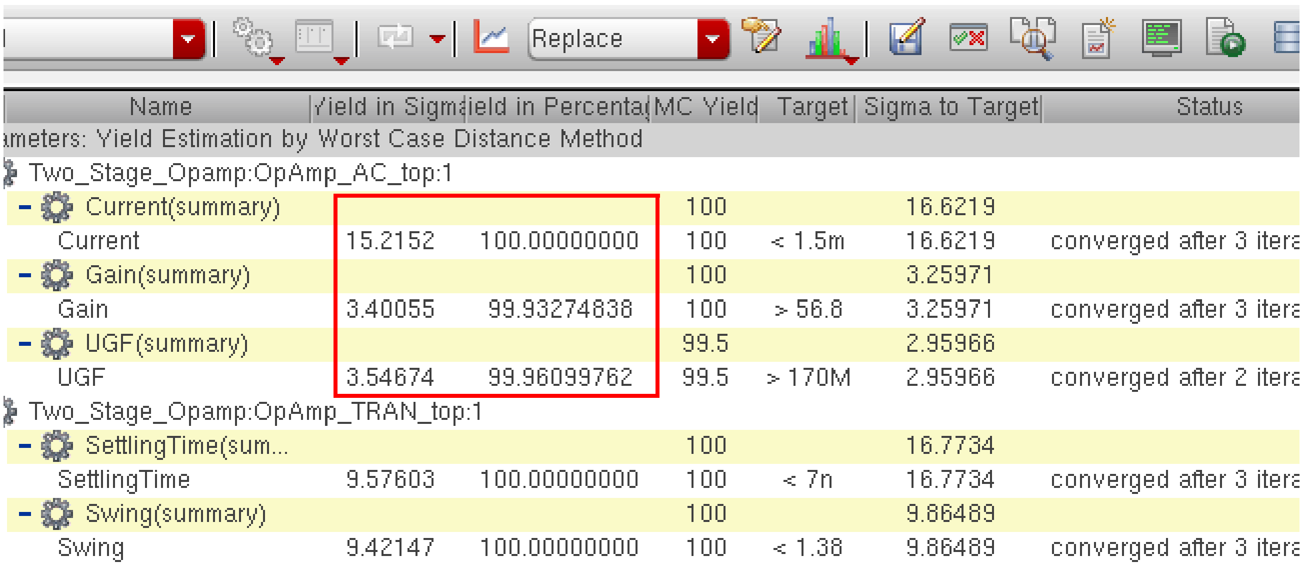


Point	Test	Output	Nominal	Spec	Weight	Min	Max	WCD_C0	WCI
Parameters: M3_fingers=10, M3_fw=11u, M3_l=1.3u, M1_fingers=20, M1_fw=7u, M1_l=900n, C0_c=349.9f, C0_m=1, C0_w=2(									
801	Two_Stage_Opamp:OpAmp_AC_top:1	Current	930.9u	< 1.5m		870.1u	1.127m	1.127m	91
801	Two_Stage_Opamp:OpAmp_AC_top:1	UGF	189.9M	> 170M		173.7M	217.9M	217.9M	18
801	Two_Stage_Opamp:OpAmp_AC_top:1	Gain	57.52	> 56.8		56.95	58.23	58.23	56
Parameters: M3_fingers=10, M3_fw=11u, M3_l=1.2u, M1_fingers=20, M1_fw=7u, M1_l=900n, C0_c=349.9f, C0_m=1, C0_w=2(									
794	Two_Stage_Opamp:OpAmp_AC_top:1	Current	930.9u	< 1.5m		870.1u	1.127m	1.127m	91
794	Two_Stage_Opamp:OpAmp_AC_top:1	UGF	194M	> 170M		177.7M	222.2M	222.2M	19
794	Two_Stage_Opamp:OpAmp_AC_top:1	Gain	57.25	> 56.8		56.7	57.98	57.98	5
Parameters: M3_fingers=8, M3_fw=15u, M3_l=1.1u, M1_fingers=14, M1_fw=9.6u, M1_l=900n, C0_c=349.9f, C0_m=1, C0_w=2									
788	Two_Stage_Opamp:OpAmp_AC_top:1	Current	1.235m	< 1.5m		1.151m	1.503m	1.503m	1.2
788	Two_Stage_Opamp:OpAmp_AC_top:1	UGF	188M	> 170M		173M	213.8M	213.8M	18
788	Two_Stage_Opamp:OpAmp_AC_top:1	Gain	57.5	> 56.8		56.88	57.99	57.99	56
Parameters: M3_fingers=8, M3_fw=14u, M3_l=1.1u, M1_fingers=14, M1_fw=9.6u, M1_l=900n, C0_c=349.9f, C0_m=1, C0_w=2									
746	Two_Stage_Opamp:OpAmp_AC_top:1	Current	1.235m	< 1.5m		1.151m	1.503m	1.503m	1.2
746	Two_Stage_Opamp:OpAmp_AC_top:1	UGF	189.6M	> 170M		174.6M	215.4M	215.4M	18
746	Two_Stage_Opamp:OpAmp_AC_top:1	Gain	57.47	> 56.8		56.82	57.95	57.95	56

From the results of an optimization run, you can identify a best design point and create a reference point. For this, right-click on the design point and choose *Create Reference Point*.

Next, you can use this reference point and run high yield estimation to verify if there is any improvement in the yield for the selected specifications. To use the reference point for high yield estimation run, ensure that the *Use Reference Point* check box on the High Yield Estimation form is selected.

For the example given above, the results show significant improvement in the yield and the worst case distance values, as shown below:



Name	Yield in Sigm	ield in Percenta	MC Yield	Target	Sigma to Target	Status
ameters: Yield Estimation by Worst Case Distance Method						
Two_Stage_Opamp:OpAmp_AC_top:1						
Current(summary)			100		16.6219	
Current	15.2152	100.00000000	100	< 1.5m	16.6219	converged after 3 itera
Gain(summary)			100		3.25971	
Gain	3.40055	99.93274838	100	> 56.8	3.25971	converged after 3 itera
UGF(summary)			99.5		2.95966	
UGF	3.54674	99.96099762	99.5	> 170M	2.95966	converged after 2 itera
Two_Stage_Opamp:OpAmp_TRAN_top:1						
SettlingTime(sum...			100		16.7734	
SettlingTime	9.57603	100.00000000	100	< 7n	16.7734	converged after 3 itera
Swing(summary)			100		9.86489	
Swing	9.42147	100.00000000	100	< 1.38	9.86489	converged after 3 itera

You can see that the yield in sigma value has increased for all Current, Gain, and UGF as compared to the results of the previous high yield estimation run.

6

Monte Carlo Post-Processing

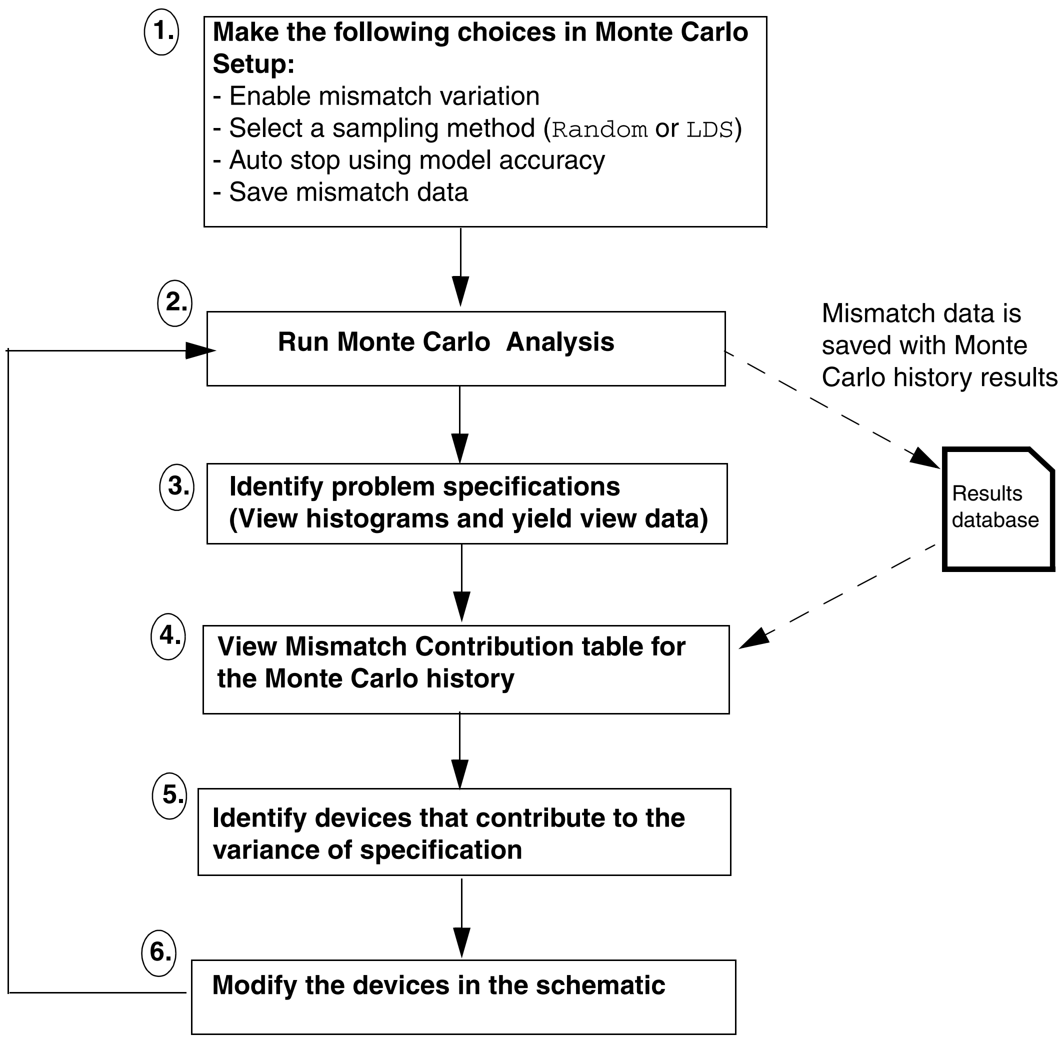
After the Monte Carlo simulation run is complete, you can run the Monte Carlo post-processing features to further analyze and optimize the design. This chapter describes these features and explains how you can use them to optimize the design.

This chapter covers the following topics:

Analyzing the Mismatch Contribution

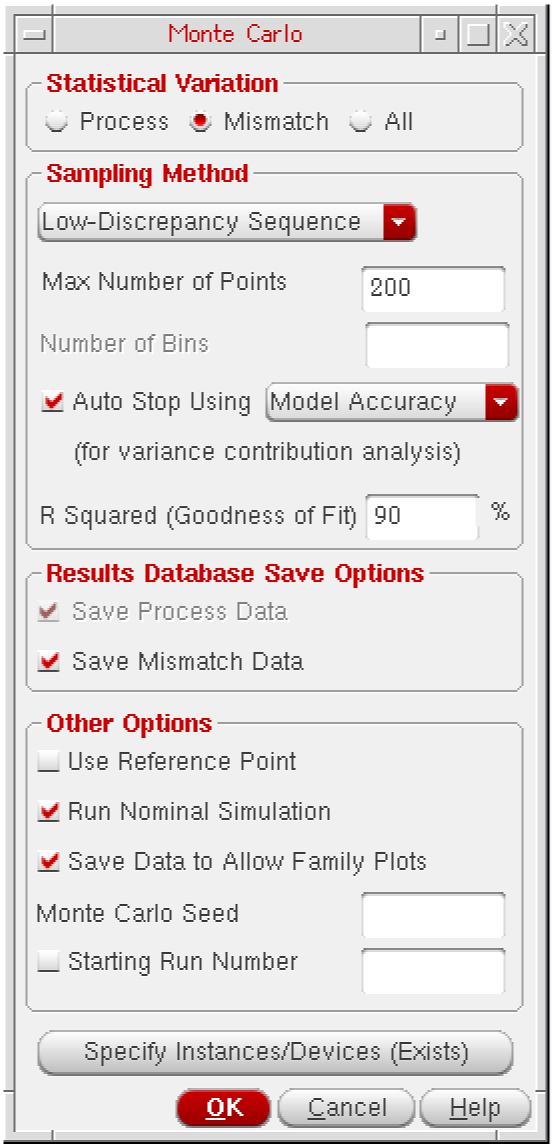
Mismatch Contribution analysis is a Monte Carlo post-processing feature that helps in identifying the important contributors to mismatch variation. You can then modify the identified devices in the schematic and make the design less sensitive to mismatch variation.

The following illustration describes how you can save and analyze the mismatch contribution data to improve your design.



Setting up Monte Carlo Run

You can set up Monte Carlo run options using the Monte Carlo form. As mentioned in step 1 above, you need to set the run options highlighted in the form shown below:



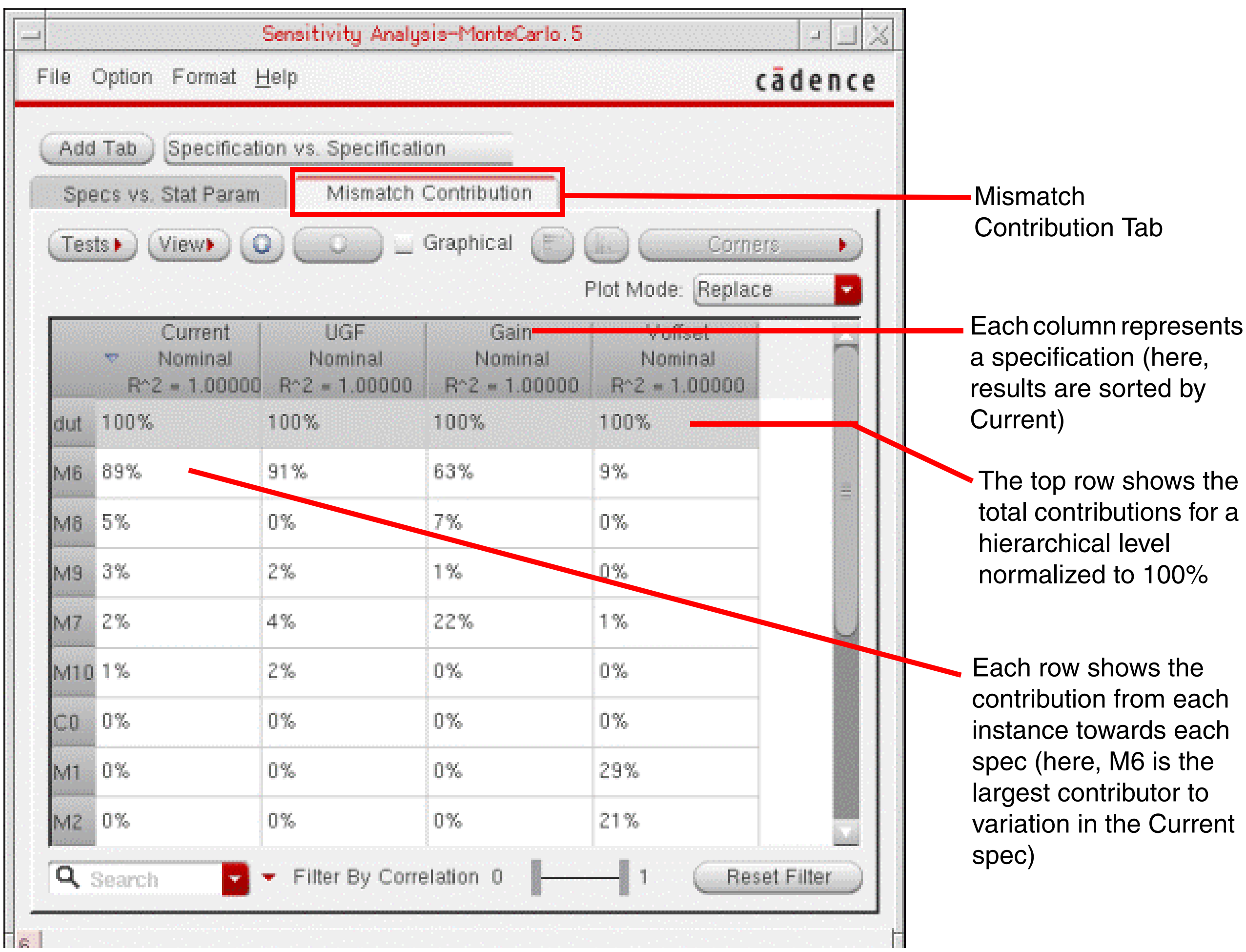
For more details about setting up Monte Carlo to automatically stop the simulation based on the accuracy of the modeling of variation in the outputs, see the *Auto Stop Using* checkbox in the Monte Carlo form as described in the *Virtuoso Analog Design Environment XL User Guide*.

Viewing Mismatch Contribution Results

You can view the important contributors to mismatch variation by using the *Mismatch Contribution* post-processing command that is run on the Monte Carlo results. For this, do one of the following:

- Click *Mismatch Contribution* on the toolbar in the Results tab.
- In the History tab on the Data View pane, right-click on a Monte Carlo history item and choose *Mismatch Contribution*.

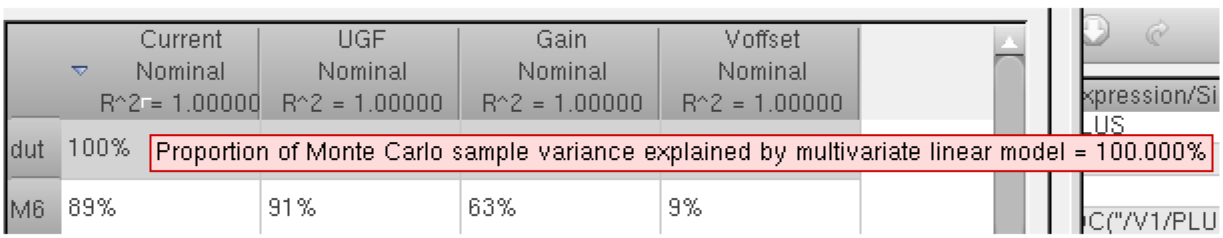
The Sensitivity Analysis window is displayed showing the *Mismatch Contribution* tab, as shown below.



In the example shown above, each row in the mismatch contribution table represents an instance of the design and each column represents the output specifications. Each cell shows the contribution (in percentage) of an instance towards the variance of the corresponding specification. Note that all the contributions are normalized to 100%.

Important

By default, ADE XL computes the proportions of a Monte Carlo sample variation by using a linear model. This is indicated by the `R^2` (R squared) value in the column headers. The tooltip for column headers also shows how these proportions are calculated.



If the `R^2` value is lower than the threshold value of 90%, the results of the linear model might not be useful. Therefore, in such cases, ADE XL automatically switches to the quadratic model to calculate variance data for a particular specification more accurately. Note that the column header is also updated to indicate this change.

The following topics show how to modify the data view in the Mismatch Contribution table:

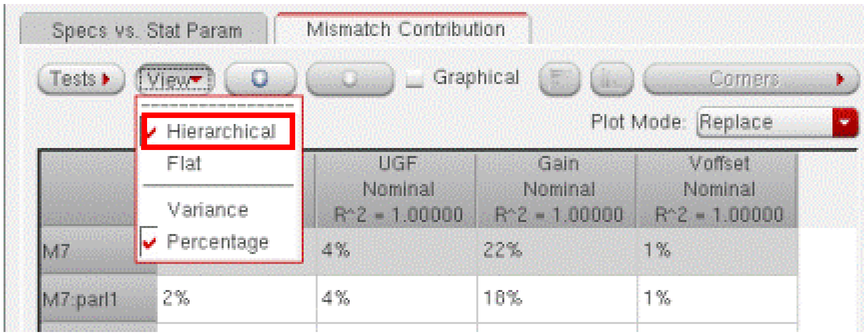
- Switching Between the Flat View and Hierarchical View
- Navigating Through the Hierarchy Levels
- Changing the Test
- Switching Between Percentage and Variance Formats
- Sorting Data
- Highlighting a Device on Schematic

Switching Between the Flat View and Hierarchical View

By default, the table shows the flat view in which the data is shown on a parameter basis. However, you can also choose to view the data in the hierarchical view in which the mismatch contribution data is displayed on a per instance basis, where the contribution attributed to an instance is the sum of the individual mismatch parameter variance.

To view the hierarchical view, do the following:

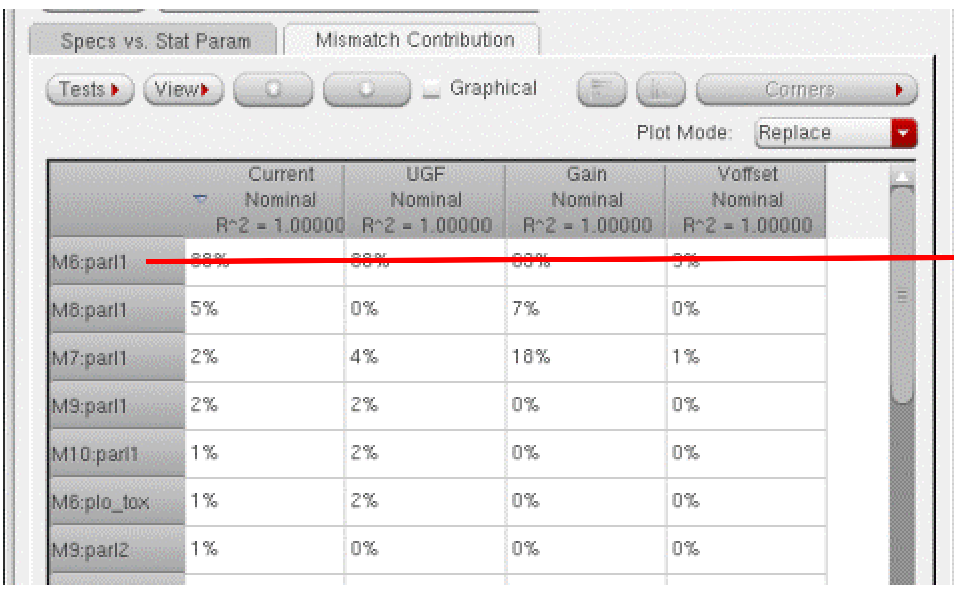
➡ Click View given at the top of the tab and choose `Hierarchical`.



Similarly, you can select `Flat` to switch from the hierarchical view to the flat view.

Note: If you delete the simulation data, the hierarchical view will not be available.

In the flat view, each row shows the individual statistical parameter in the *<instance-name>:<parameter-name>* format. As there can be multiple parameters for a device, each parameter is shown in a separate row, as shown in the example below.

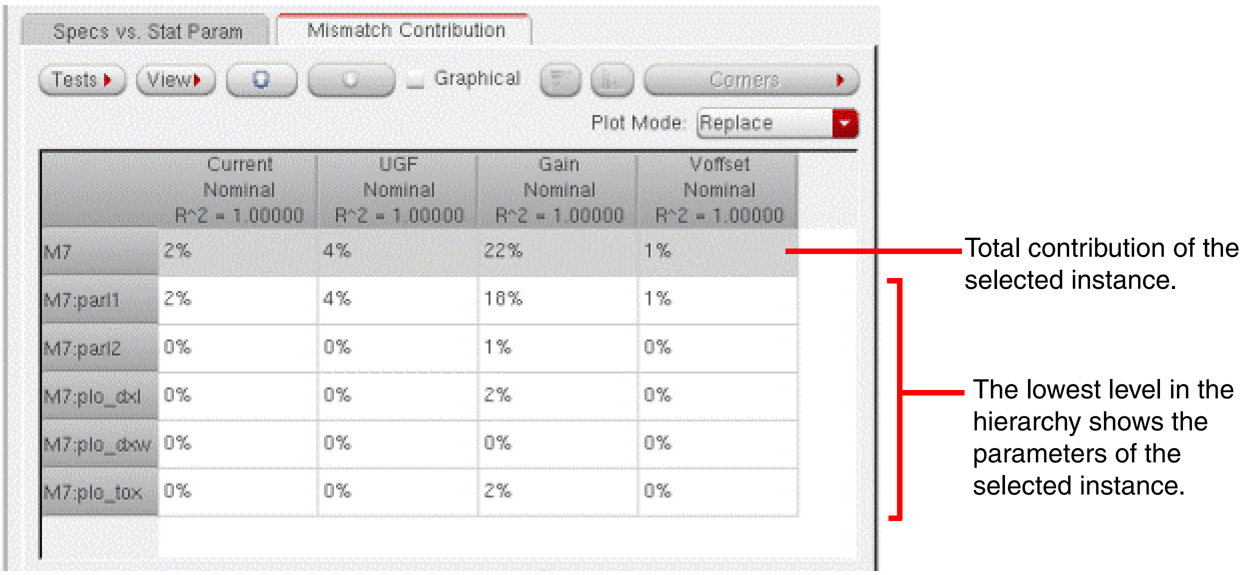


<Instance-name>:<parameter-name>

Navigating Through the Hierarchy Levels

In the hierarchical view the top level is displayed first. You can use the up and down arrow buttons ( ) given at the top of the tab to traverse through the hierarchy. Alternatively, you can double-click in a cell to navigate to the lower levels in the hierarchy.

The lowest level in the hierarchy shows the parameters for a given instance, as shown in the example below.



Changing the Test

The mismatch contribution table can display data for only one test at a time. By default, it shows the data for the first test in the ADE XL setup.

To switch the view to show data for another test, do the following:

➡ Click Tests at the top of the tab and choose a test name from the drop-down list.

Mismatch data for the selected test is displayed in the table.

Switching Between Percentage and Variance Formats

By default, the contribution of each device or parameter is shown in the percentage format.

To display the actual variance values, do the following:

➡ Click View at the top of the tab and choose *Variance*.

Similarly, you can select `Percentage` to switch from variance to the percentage formats.

Sorting Data

By default, the order of devices or parameters in the mismatch contribution table is the same as the order of occurrence in the design. For better data analysis, you can sort the data in a row or column using the percentage or variance values in the increasing or decreasing order.

To sort the data in a row or column, do this:

➡ Right-click the header of the row or column and choose any one of the following commands:

- ❑ *Sort by value*
- ❑ *Sort by absolute value*

Highlighting a Device on Schematic

While analyzing data in the mismatch contribution table, you can highlight a device on the schematic to view its exact location. This can help in easily identifying and modifying a device in schematic to improve the variance data, if required.

To highlight a device on schematic, do this:

➡ Right-click on a row and choose *Highlight on Schematic*.

Alternatively, you can press `H`.

Creating Statistical Corners

After running the Monte Carlo simulations, you can analyze the yield for various specifications and identify the specification for which results can be improved to optimize the design. For these identified specifications, you can then create the statistical corners that can be further used to run simulations for design optimization.

In ADE XL, you can create a statistical corner from any of the simulated samples. There are several options to create a corner including selection of a point on the histogram, creating a corner out of the worst sample or by percentile.

The methods that use a selected sample, worst sample, and percentile to create the statistical corner are available in ADE XL. For information about these methods, refer to the following topics in the *Analog Design Environment XL User Guide*:

- Creating Statistical Corner from Worst Sample
- Creating Statistical Corner from Percentile
- Creating Statistical Corner from a Selected Sample

Creating a K-Sigma Statistical Corner

All of the above-mentioned methods define the corner by using one of the simulated Monte Carlo samples. To create a 3-sigma statistical corner, such as using the worst sample you must have simulated a large number of samples. This need for multiple simulated samples can be avoided by using the K-sigma method.

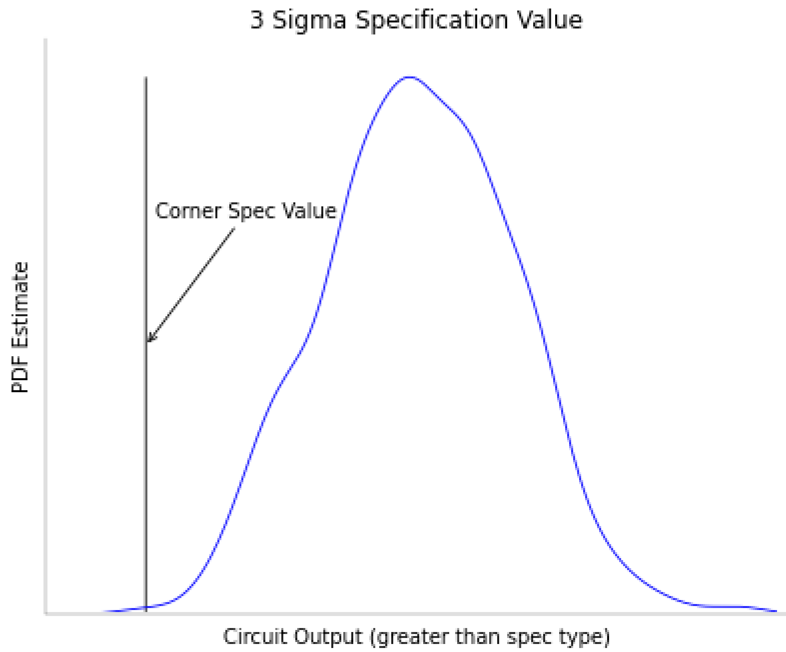
In this method, you specify a target yield-in-sigma value and a statistical corner is created that meets this specified target yield value.

The fast K-sigma corner method performs the following tasks:

- Runs the Monte Carlo Sampling with only a few hundred samples as compared to the traditional method where you were required to run thousands of these samples.
- Creates the 3-sigma statistical corner very quickly.

Note: Minimum 1 and maximum 11 extra simulations per corner are needed for this method.

The fast K-sigma corner algorithm estimates the Probability Density Function of the performance distribution maintaining accuracy for non-normal distributions. The specification target value is computed from the PDF estimate.

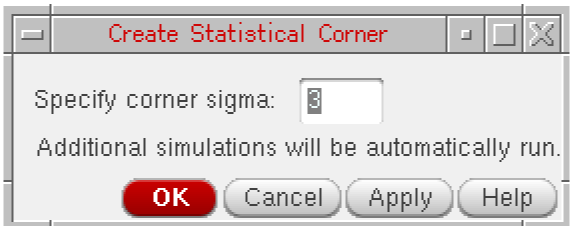


A statistical corner is then created that matches the target specification value. It is possible that multiple corners meet the target specification criteria. Therefore, the K-sigma method finds the most representative corner by calculating the minimum distance to the nominal point. This representative corner has a greater probability to occur. The statistical corner can then be used for further analysis of the design.

Perform the following steps to create the K-sigma statistical corner:

1. Open the Monte Carlo results in the *Yield* view and identify the specification for which you need to improve the yield.
2. Right-click anywhere in the row for the identified specification and choose C*reate Statistical Corner (Specify Yield in Sigma).*

The *Create Statistical Corner* form appears.

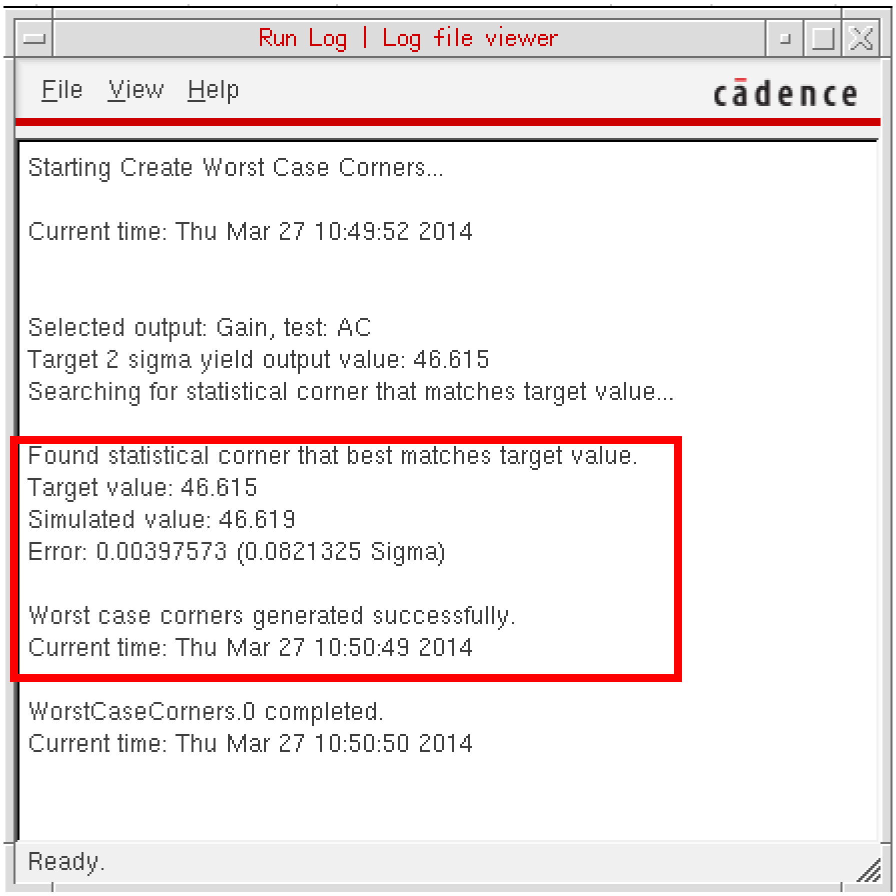


3. In this form, in the *Specify corner sigma* field, specify the sigma value. The default and maximum value is `3`.

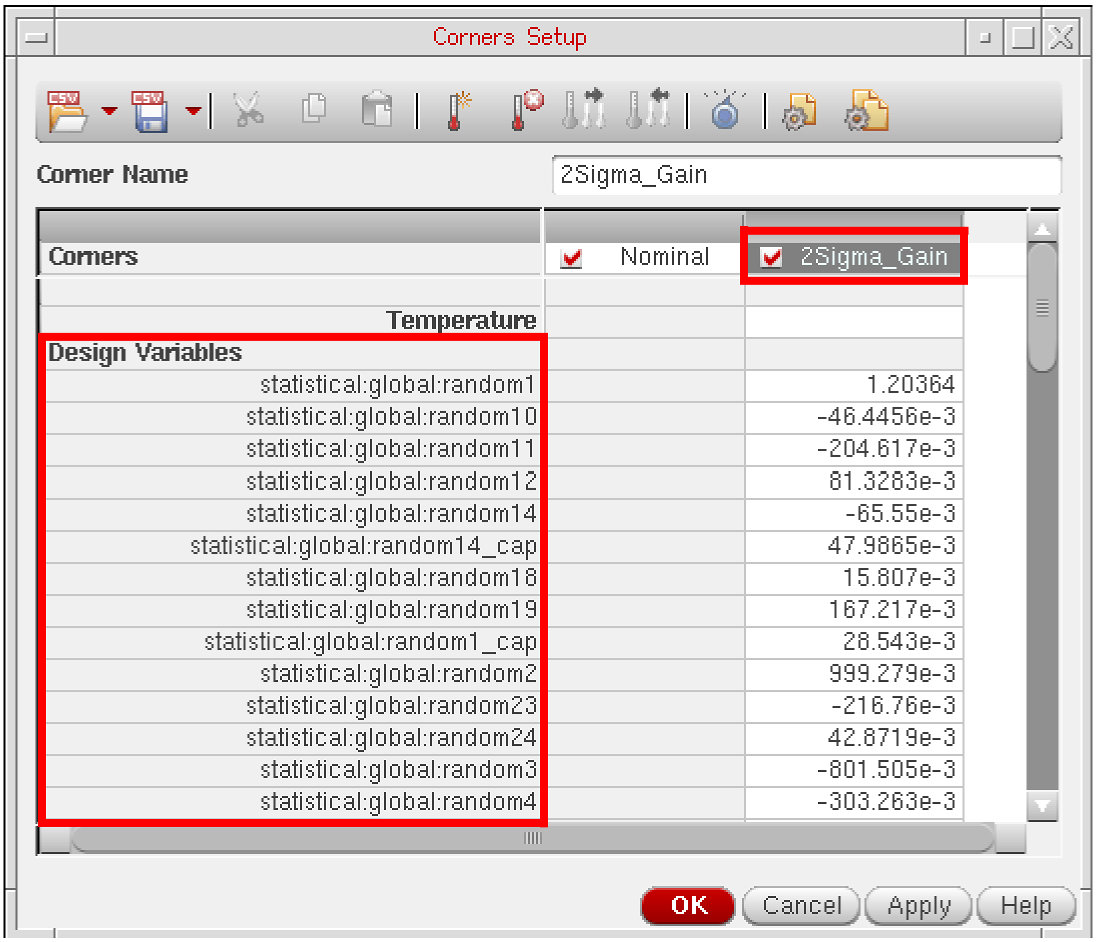
4. Click *OK*.

Note: If you want to create a statistical corner with target sigma greater than `3`, you can use high yield estimation to create the statistical corner. For more information, see Creating Statistical Corners From a Worst Case Distance Analysis in Chapter 5, "High Yield Estimation,"

An example of log file created during the K-sigma run is shown in the figure below:



After the worst case corner simulation is complete, the K-sigma corner is displayed in the *Corners Setup* form as shown in the figure below.



Note: The corner is named as `<`*`sigma_value`*`>Sigma_Gain`, where `<`*`sigma_value`*`>` is the corner sigma value you specify in the *Create Statistical Corner* form. For example, if you specify the sigma value as `2`, the statistical corner name will be `2Sigma_Gain`.

Now, you can simulate the design over the generated statistical corner for further analysis or design tuning.

7

Multi-Technology Simulation

You can use multi-technology simulation (MTS) when designing high-speed interfaces between integrated circuits (ICs) to see the effects of signals traveling between two or more chips. You can also use MTS for custom IC system-in-package (SiP) designs. SiP designs are fully functional systems or subsystems in an IC package. MTS allows you to combine passive components with ICs fabricated using different process technologies while minimizing parasitic loading.

Important

In order to run a simulation in multi-technology mode, your design must have a design configuration view. This requirement makes it possible for you to configure MTS blocks from the hierarchy editor (HED).

Your MTS design should consist of a top-level schematic that contains two or more blocks, each with its own associated set of process definitions, model libraries, simulation states, and other setup files. You cannot use MTS with a non-hierarchical schematic because each different technology must be self-contained. You must instantiate device instances from different technologies in different cell views so that the program can map model information appropriately.

To specify MTS, you need to do the following:

- Enable multi-technology simulation on the Choosing Simulator form.
- Specify MTS options

See also

- Specifying Test and Block Information when Adding Model Files to Corners on page 211
- Disabling MTS for Third-Party Simulators on page 211

Enabling Multi-Technology Simulation

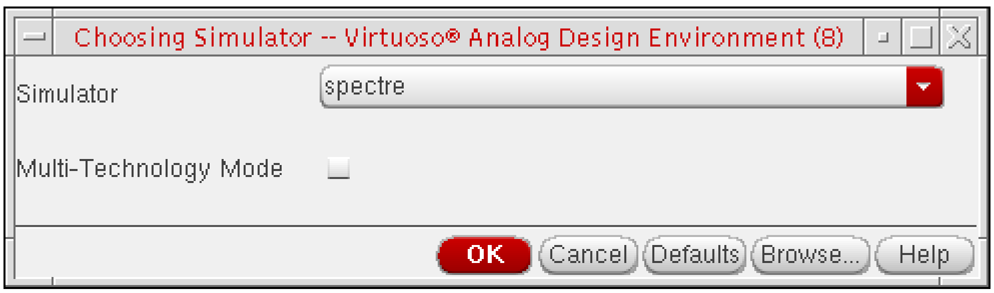
Important

In order to run a simulation in multi-technology mode, your design must have a design configuration view.

To enable multi-technology simulation, do the following:

1. On the Data View pane, right-click the test or analysis name and select *Simulator*.

   The Choosing Simulator form appears.



2. Mark the *Multi-Technology Mode* check box.
3. Click *OK*.

   You can now specify MTS options.

Specifying MTS Options

Important

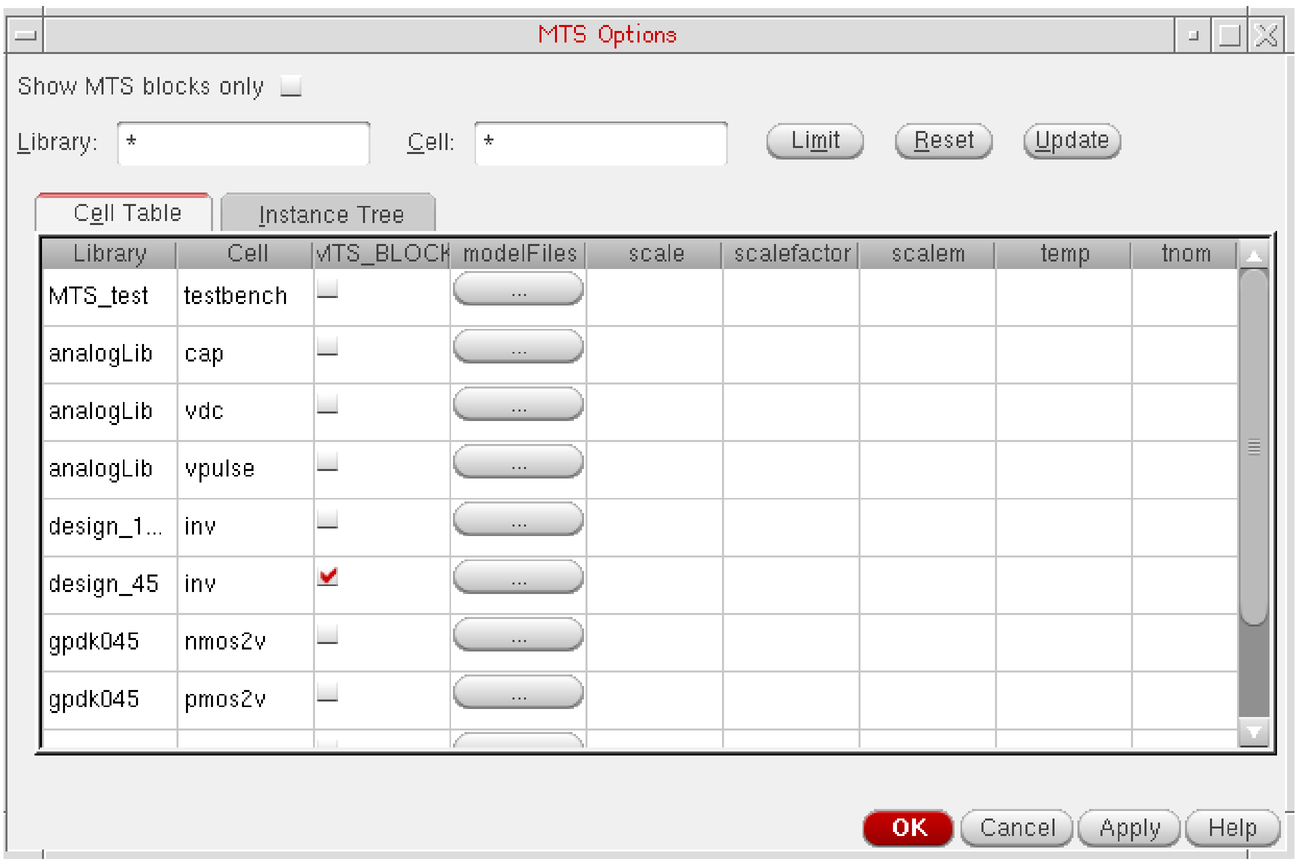
Before specifying MTS options, you must enable MTS.

To specify MTS options, do the following:

➤ On the Tests and Analyses pane, right-click the test or analysis name and select *MTS Options*.

Note: The test or analysis you right-click must be one for which you have enabled MTS.

The MTS Options form appears.



MTS blocks are indicated by a check mark in the *MTS_BLOCK* column.
You can specify settings that apply to particular cells or instances on this form.

See the following topics for more information:

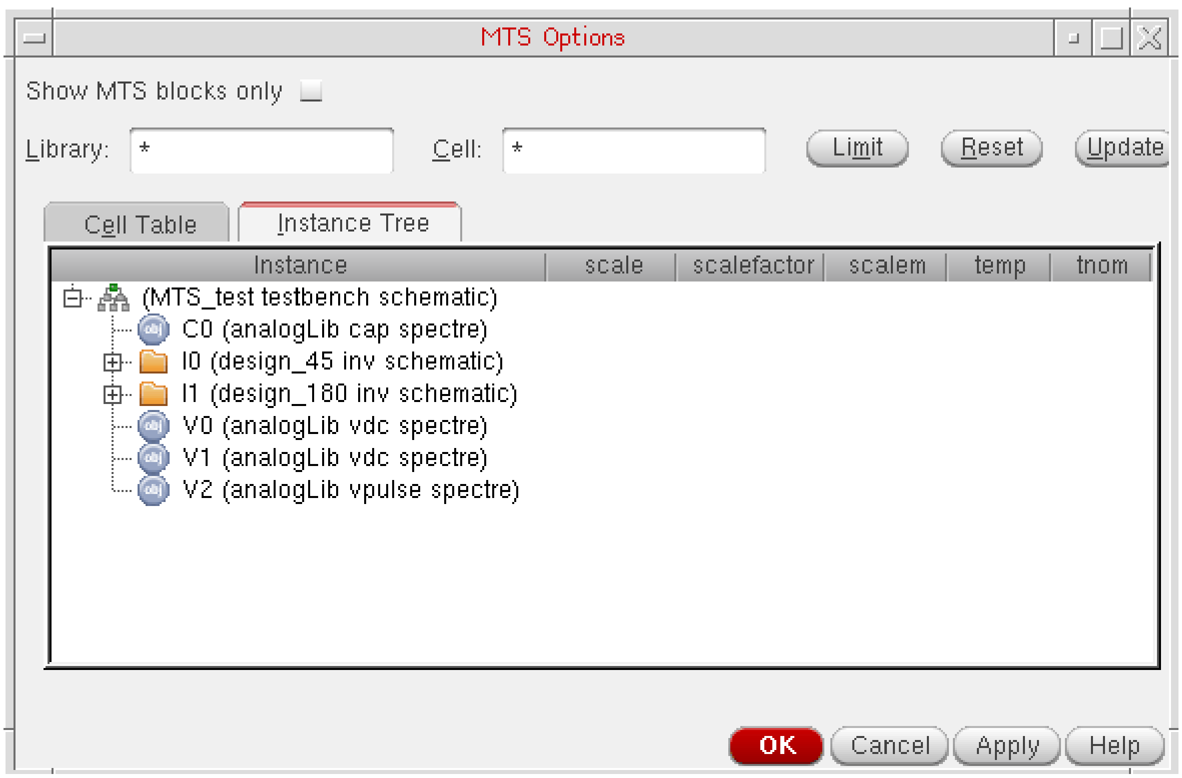
- Viewing Cells by Instance Name on page 208

Viewing Cells by Instance Name

To view cells by instance name, do the following:

➤ On the *MTS Options* form, select the *Instance Tree* tab.

Instance names appear alphabetically in a tree view on the form.



To restore the cell table, do the following:

➤ Select the *Cell Table* tab.

The cell table view appears on the form.

See also "Limiting the View by Library or Instance" on page 209.

Limiting the View by Library or Instance

You can limit what appears on the MTS Options form by library name, cell name, or both.

Note: You can reset the filter to view all cells in all libraries by clicking *Reset*.

To limit the view of design components on the MTS Options form to only those in a particular library or set of libraries, do the following:

1. In the *Library* field, type a filtering string.
2. Click *Limit*.

   Only those design libraries that match the filtering string appear on the MTS Options form.

To limit the view of design components on the MTS Options form to a particular cell or set of cells, do the following:

1. In the *Cell* field, type a filtering string.
2. Click *Limit*.

   Only those cells that match the filtering string appear on the MTS Options form.

To limit the view by library name and by cell name, do the following:

1. In the *Library* field, type a filtering string.
2. In the *Cell* field, type a filtering string.
3. Click *Limit*.

   Only those cells that match the filtering string in the *Cell* field contained in those libraries that match the filtering string in the *Library* field appear on the MTS Options form.

Specifying a Cell as an MTS Block

To specify a cell as an MTS block, do the following:

➤ In the row for that cell, mark the *MTS_BLOCK* check box.

The library cell you selected is enabled for multi-technology simulation.

If you open the Model Library Setup form, you will see a new model library tree for that library and cell so that you can specify model libraries specific to that MTS block.

Specifying Simulator Options for an MTS Block

To specify simulator options (such as *modelFile*, *tnom*, *scale*, *scalefactor*, *temp*, or *scalem*) for a particular cell or instance that you have specified as an MTS block, do the following:

➤ Click in the column for the simulator option you want to specify and type a value.

The value you specify applies to the simulation of that MTS block.

Note: You can also specify a `VAR` expression for the *modelFile*, *scale*, *scalem*, *scalefactor*, *temp*, and *tnom* columns in the *MTS Options* form. The variable name specified in the `VAR` expression is added to the *Global Variables* list on the *Data View* assistant pane and *Design Variables* section of test.

Specifying a Model Library for an MTS Block

To specify a model library for an MTS block, do the following:

➤ In the row for the cell that you have specified as an MTS block, click the button in the *modelFiles* column.

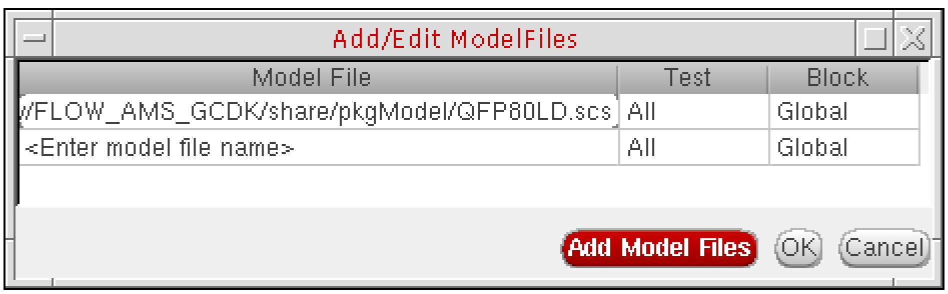
The Model Library Setup form appears. See "Specifying Model Libraries" in the *Virtuoso Analog Design Environment XL User Guide* for more information about using this form.

Specifying Test and Block Information when Adding Model Files to Corners

To specify a particular test to which you want to apply a model file or a particular section of a model file when adding model files for corners analysis, do the following:

1. Follow the instructions for adding model files to corners but do not click *OK*.

   The model file you added appears on the Add/Edit Model Files form.



2. In the *Test* column, double-click and, from the drop-down menu, select a particular test for which you want to use the model file.

   By default, the program uses the model file for all tests.

3. In the *Block* column, double-click and, from the drop-down menu, select a particular MTS block to which you want to apply the model file.

   By default, the program uses the model library for the entire design: *Global*.

4. Click *OK*.

Disabling MTS for Third-Party Simulators

To disable the MTS feature for a third-party simulator that does not support it, do the following to overload the method:

➤ Change

```
defmethod( asiIsMTSSupported ( ( session simulation_session ) )
    t
)
```

to

```
defmethod( asiIsMTSSupported ( ( session className ) )
    nil
)
```

where, `className` is the name of the third-party simulator class derived from `simulation_session`.

Detecting Context Symbol/Name Collisions

As a context developer, you need to check for conflicts that arise when you define a function or variable in more than one context (name collisions).

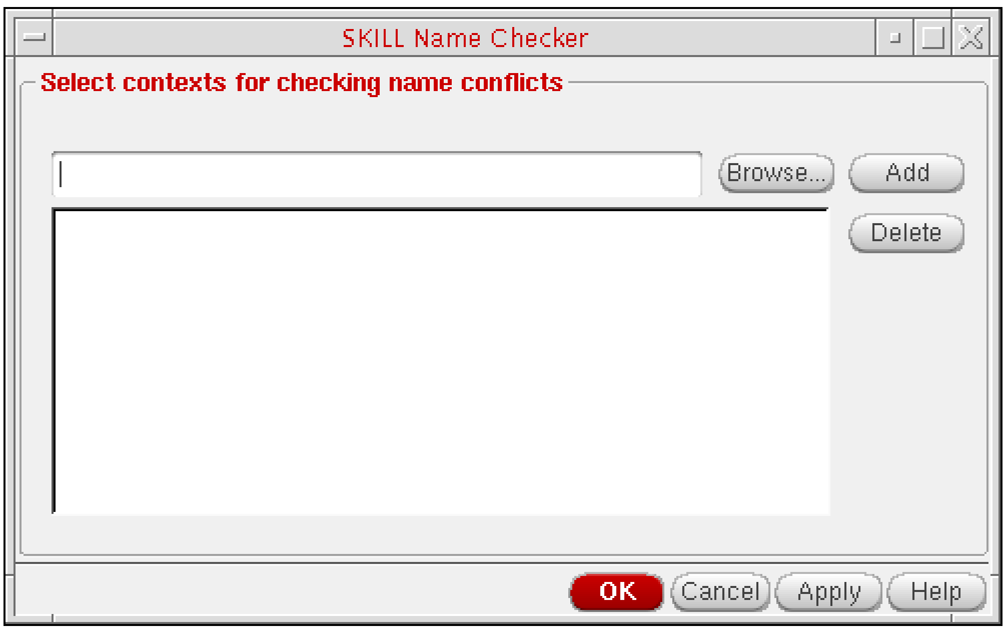
To check for name collisions in loaded contexts, do the following:

1. In the CIW, choose *Tools – Conversion Tool Box*.

   The Conversion Tool Box appears.

2. Click *Check SKILL Name Conflicts*.

   The SKILL Name Checker form appears.



3. Click *Browse*.

   A browser window appears.

4. For each context file you want to add to the check, do the following:

   a. Navigate to and select a valid context file and click *Apply*.

      The file name appears in the field on the SKILL Name Checker form.

   b. On the SKILL Name Checker form, click *Add*.

      The file name moves from the field to the list area.

5. On the SKILL Name Checker form, click *OK*.

Duplicate definitions appear in the SKILL Name Checker Results window as follows:

```
symbol 'functionName already defined by context "contextName"
```

where `functionName` is the name of the function and `contextName` is the first context in which the program found `functionName`.

8

Environment Variables

This appendix describes public environment variables that control characteristics of the Analog Design Environment GXL (ADE GXL). You can customize the operation and behavior of ADE GXL features by changing the values of particular environment variables.

Also see:

"Environment Variables" in the *Virtuoso Analog Design Environment XL User Guide*

ADE GXL Environment Variables

You can set the following environment variables in your `.cdsenv` or `.cdsinit` files to customize the settings for simulations or results:

- sensitivityPlotContinuousLine
- sensitivityThumbnailMaxPointsForLinePlot
- digitsToShowForYieldInPercentage
- sortVariablesOpt
- stopManualTuningOnSessionExit
- useDoubleSidedSigma

sensitivityPlotContinuousLine

Specifies if a line plot should be plotted for the results of sensitivity analysis.

In `.cdsenv`:

```
adexl.plotting sensitivityPlotContinuousLine boolean t
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.plotting" "sensitivityPlotContinuousLine" 'boolean
     t)
```

Valid Values:

`t`	This is the default value.
`nil`	

sensitivityThumbnailMaxPointsForLinePlot

Specifies the minimum number of points for which line plots should be plotted in the thumbnails in the Sensitivity Analysis results window. When the number of points is more than the value of this variable, the thumbnail plots show scatter plots.

In `.cdsenv`:

```
adexl.plotting sensitivityThumbnailMaxPointsForLinePlot int t
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.plotting"
     "sensitivityThumbnailMaxPointsForLinePlot" 'int t)
```

Valid Values:

A positive integer value

Default Value: `10`

digitsToShowForYieldInPercentage

Specifies the number of digits to be displayed for values in the *Yield In Percentage* column on the Results tab for High Yield Estimation run.

In `.cdsenv`:

```
adexl.gui digitsToShowForYieldInPercentage int 6
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.gui" "digitsToShowForYieldInPercentage" 'int t)
```

Valid Values:

A positive integer value

Default Value: `10`

sortVariablesOpt

Specifies if the variables and parameters should be sorted before generating random samples for an optimization run. By default, the variables are not sorted before the run is started. However, you can sort them by setting this variable to `t` so as to ensure that the result of different optimization runs is same irrespective of the order of variables and parameters.

In `.cdsenv`:

```
adexl.algorithm sortVariablesOpt boolean nil
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.algorithm" "sortVariablesOpt" 'boolean t)
```

Valid Values:

`t`	Sorts the variables and parameters before generating random samples.
`nil`	Does not sort the variables and parameters before generating random samples. This is the default value.

stopManualTuningOnSessionExit

Specifies if the Manual Tuning run mode should be stopped when ADE XL GUI is closed while that run is in progress.

In `.cdsenv`:

```
adexl.simulation stopManualTuningOnSessionExit boolean nil
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.simulation" "stopManualTuningOnSessionExit"
     'boolean t)
```

Valid Values:

`t`	Stops the currently running Manual Tuning run when the ADE XL GUI is closed. In the next session, the Run Simulation button is green in color and you can either start a new Manual Tuning run or submit new points in the previous Manual Tuning run.
`nil`	Does not stop the currently running Manual Tuning run when the ADE XL GUI is closed. When you open a new ADE XL session, the Run Simulation button is yellow in color and you can continue with the same Manual Tuning run that was running earlier. This is the default value.

useDoubleSidedSigma

Specifies whether the K-sigma statistical corner is single-sided or double-sided. You can set this variable while creating K-sigma statistical corner from the following methods:

- K-sigma corner from Monte Carlo

- ❑ K-sigma corner from High Yield Estimation

In `.cdsenv`:

```
adexl.algorithm useDoubleSidedSigma boolean nil nil
```

In `.cdsinit` or the CIW:

```
envSetVal("adexl.algorithm" "useDoubleSidedSigma" 'boolean nil)
```

Valid Values:

`t`	When this variable is set to `t`, the double-sided K-sigma statistical corner is created.
`nil`	When this variable is set to `nil`, the single-sided K-sigma statistical corner is created. This is the default value.

Index

A

C

D

F

G

I

L

M

O

P