Model Editor Help

Product Version 23.1
September 2023

Contents

Introducing Model Editor

See the following topics to start working with the Model Editor:

Overview of the Model Editor

You can use the Model Editor to determine the model parameters for many standard device types, such as bipolar transistors, and to create the subcircuit models for more complex device types, such as operational amplifiers. You can create accurate models and parts for off-the-shelf devices by converting information from the device manufacturer's data sheet into simulation model[1] parameter values, without taking measurements of a real device. Then you can include the new, custom device model as part of your own model library.

You may want to use a model based on typical values from the data sheet for most of your simulation work, then create best/worst-case models for checking your design.

The advantage of modeling devices rather than merely using their data sheet values is that for more complex devices, especially semiconductor devices, the physical model for transistor behavior views the transistor from the inside, while the manufacturer’s measurements show

1. Through out this manual simulation model refers to PSpice models and PSpice Simulator models

the transistor behavior from the outside. Converting data sheet values to physical model parameters produces more accurate simulation[1] results.

Simpler devices, such as resistors, may only need the resistance value to have a complete model.

The Model Editor closely approximates each of the device characteristics from the most relevant subset of the model parameters. Some of these curves will not match precisely the results of the simulation.

For some of the device types supported by the Model Editor, you can create models based on simulator[2] provided templates. These models are referred to as parameterized models or template-based simulation models.

Getting started with the Model Editor

The Model Editor has an all-new interface. See Using the Model Editor interface for more information.

What the Model Editor does

The Model Editor converts data sheet information into device model (.MODEL) and subcircuit (.SUBCKT) definitions and saves these definitions to model files that simulator can search when looking for simulation models.

- See Device types supported by the Model Editor for a list of model types supported by the Model Editor.
- See Parameterized Models supported by the Model Editor for a list of template-based model types supported by the Model Editor.
- See Saving Global Models (and Parts) for information on saving and configuring the models and parts you create in the Model Editor.

Ways to use the Model Editor

Use Model Editor when you want to derive models from data sheet curves provided by manufacturers.

1. Throughout this manual simulation refers to simulation in PSpice and PSpice Simulator.
2. Throughout this manual simulator refers to PSpice A/D or PSpice Simulator

Using the Model Editor alone

When you do not want the model you are creating to be tied to a part instance on your schematic page or to a part-editing session:

- ❑ Define a new model, and then automatically create a part. Any new models and circuits are automatically available to any design. See Part creation options for more information.
- ❑ Define a new model only (no part). You can disable the part creation feature for new models. The model definition is available to any design by changing the model reference on the part instance.
- ❑ Examine or verify the electrical characteristics of a model without running the simulator.

Using the Model Editor from within Capture's part editor

When you want to base a new model on an existing part, or to edit the model for an existing part to affect all designs that use that part:

- Edit the model definition linked to the part definition in the part library, after having loaded or created a part. Capture automatically links the new model definition created by the Model Editor to the part definition. Refer to your simulator user's guide for more information.

Using the Model Editor from within design entry tool[1]'s schematic page editor

In Capture:

When you want to test behavioral variations on a part, refine a model before making it available to all designs, or view model characteristics for a part:

- Edit a model definition for a part instance in your design, after having selected one or more part instances on the schematic page. Capture automatically links the new model definition created by the Model Editor to the selected part instances. Refer to the simulator User's Guide for more information.

1. In this guide, design entry tool is used for both OrCAD Capture and Design Entry HDL. Any differences between the two tools is mentioned, if necessary.

In Design Entry HDL:

When you want to test behavioral variations on a part, refine a model before making it available to all designs, or view model characteristics for a part:

- Edit a model definition for a part instance in your design, after having selected a part instance on the schematic page. If you create a new model definition, and want to use it for the part instance, you have to specify the new model name as the value of the IMPLEMENTATION property for the part instance. Refer to the PSpice User Guide for more information.

What the Model Editor does not support

The Model Editor does not support the following subcircuit constructs:

- optional nodes construct, OPTIONAL:
- variable parameters construct, PARAMS:
- local .PARAM command
- local .FUNC command

To refine the subcircuit definition for these constructs, use the model editor in a design entry tool.

Device types supported by the Model Editor

This device type...	Uses this definition form...	And this name prefix...
diode	.MODEL	D
bipolar transistor	.MODEL	Q
IGBT	.MODEL	Z
JFET	.MODEL	J
MOSFET	.MODEL	M
operational amplifier	.SUBCKT	X
voltage comparator	.SUBCKT	X
nonlinear magnetic core	.MODEL	K

This device type...	Uses this definition form...	And this name prefix...
voltage regulator	.SUBCKT	X
voltage reference	.SUBCKT	X
Darlington transistor	.SUBCKT	X

For more information on these device types, refer to the Analog devices chapter of the online simulator A/D Reference Manual.

Using the Model Editor interface

In the Edit Model view, the Model Editor workspace is divided into following three windows:

- a Models List docking window on the left,
- a Simulation/Smoke parameter window on the right (the main window in the workspace),
- and a Model Text docking window on the bottom.

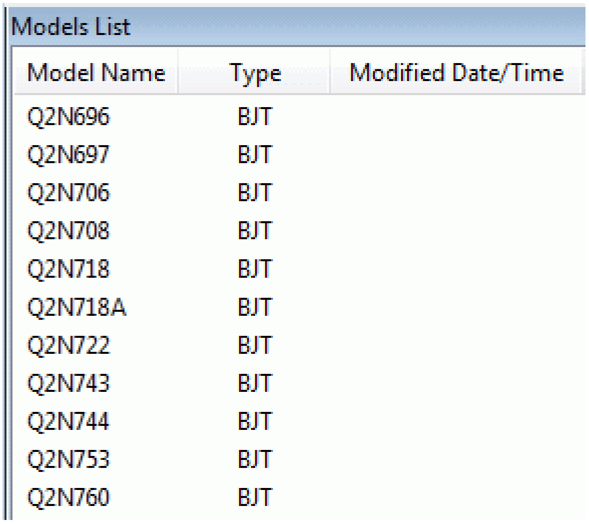
You can move the dockable windows from their default locations to the desired locations.

In the Extract Model view, the workspace of the Model Editor is divided into three windows:

- a Models List docking window on the left,
- a Spec Entry document window on the right (the main window in the workspace),
- and a Parameter docking window on the bottom.

These are only the default locations. Split bars that can be moved to enlarge, reduce, or hide the viewing area of each separate the windows. You can dock the Models List and Parameter windows to any side of the workspace. Also, you can enable or disable the display of toolbars, the status bar, and the windows by selecting the items on the View menu.

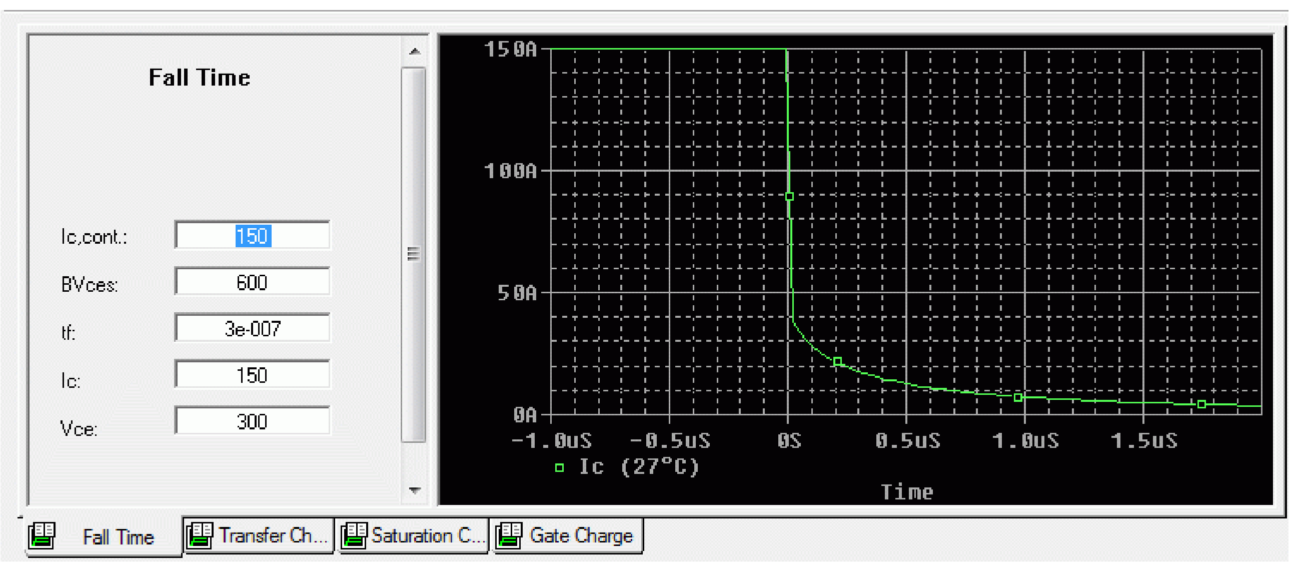
Models List docking window


Models List

Model Name	Type	Modified Date/Time
Q2N696	BJT	
Q2N697	BJT	
Q2N706	BJT	
Q2N708	BJT	
Q2N718	BJT	
Q2N718A	BJT	
Q2N722	BJT	
Q2N743	BJT	
Q2N744	BJT	
Q2N753	BJT	
Q2N760	BJT	

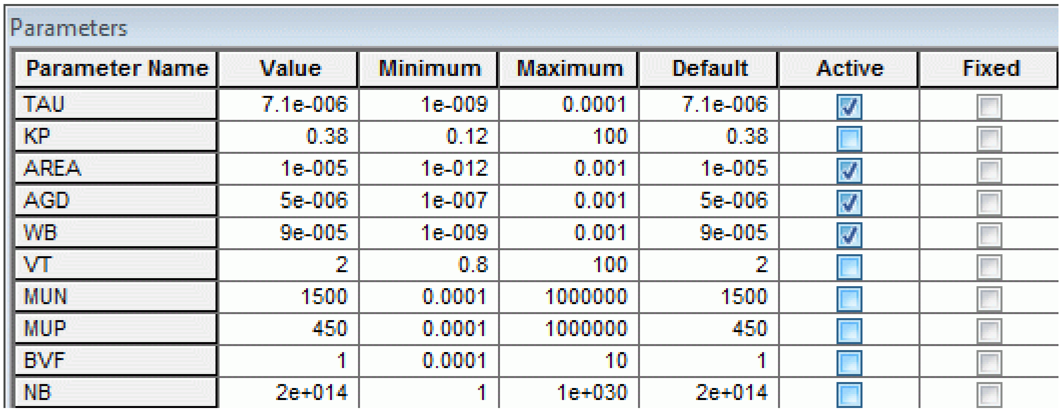
SeeUsing the Models List Window for more information.

Spec Entry document window



See Using the Spec Entry Frame for more information.

Parameter docking window



Parameters

Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
TAU	7.1e-006	1e-009	0.0001	7.1e-006	☑	☐
KP	0.38	0.12	100	0.38	☐	☐
AREA	1e-005	1e-012	0.001	1e-005	☑	☐
AGD	5e-006	1e-007	0.001	5e-006	☑	☐
WB	9e-005	1e-009	0.001	9e-005	☑	☐
VT	2	0.8	100	2	☐	☐
MUN	1500	0.0001	1000000	1500	☐	☐
MUP	450	0.0001	1000000	450	☐	☐
BVF	1	0.0001	10	1	☐	☐
NB	2e+014	1	1e+030	2e+014	☐	☐

SeeUsing the Parameter Window for more information.

Keyboard shortcuts, toolbar buttons, and menu commands

For a list of keyboard shortcuts, see Model Editor keyboard shortcuts. For descriptions of toolbar buttons and menu commands, see Model Editor menu commands.

You can customize the keyboard shortcuts and toolbar buttons. See Customizing Keyboard Shortcuts or Customizing Toolbars for more information.

Model Editor keyboard shortcuts

You can use the following key combinations as shortcuts for the corresponding menu commands.

These shortcuts are saved in the default settings. For information on creating or changing keyboard shortcut settings, see Customizing Keyboard Shortcuts.

Shortcut	Menu command
Ctrl+O	Open (library)
Ctrl+S	Save (models and library)
Ctrl+P	Print (graph)
Alt+F4	Exit
Ctrl+C	Copy (text)

Shortcut	Menu command
F	View Fit
A	View Area
P	View Previous
I	View In
O	View Out
LLLL	Redraw (the screen)
Alt M, T	IBIS Translator
Del	Delete Trace
F1	Help

Model Editor menus

Use the Model Editor commands to create and edit models from an ASCII library, enter and modify specifications, view graphs, and extract model parameters.

For information on configuring toolbar settings, see Customizing Toolbars.

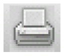
`This menu`	Performs these functions	And has these toolbar buttons
`File`	Open libraries.	
	Save library and its models.	
	Print specs.	
	Encrypt Library	
`Edit`	Cut, copy, paste, and delete text.	
`View`	Change the graph viewing scale.	
	Redraw the screen.	
	Enable or disable the display of windows and the toolbar.	

`Model`	Create, copy, and delete models. Import and export models from and to files.	
`Plot`	Add and delete traces. Modify axis settings.	
`Tools`	Extract model parameters. Customize toolbar and keyboard shortcuts. Set up the automatic part creation option. Enable or disable graph splitter synchronization.	
`Window`	Arrange Spec Entry windows.	
`Help`	Access online help on all aspects of the Model Editor.	

File menu

Use the commands on the File menu to open and save libraries, establish a new workspace, and print specs and plots.

`Command`	Description
`New`	Create a new model library.
`Open` 	Open a model library.
`Close`	Close the current model library.
`Save`	Save the current data and model parameters to the open library.
`Save As`	Save the entire library as a new library, or overwrite an existing library.

Print 	Print a copy of any or all of your plot windows. Choose Print, and then select one or more plot windows to print. Use the Select All button to select all of the plot windows.
Print Preview 	Preview the specs and plots before you print them.
Page Setup	Set page size and printing options.
Export to <design entry tool> Part Library	Run translator to create symbols for the indicated front-end. Symbols are created for all the parts in the specified library.
Encrypt Library 	Launch the Library Encryption dialog box. You can specify the library to be encrypted and whether to encrypt a library completely or partially from the Library Encryption dialog box. You can also specify the folder where the encrypted library should be placed.
Model Import Wizard [design entry tool]	Launch Model Import Wizard to create/associate part symbols for the indicated front-end. Symbols are created for all the models in the specified library. This is an interactive mode of part creation. Users can view and if required change the symbol shapes associated with a simulation model before the changes are saved in the symbol library (.olb).
1 2 3 ...	Shortcuts for opening any of the four most recently used libraries. The file name listed by 1 is the most recently used file, and the file name listed by the highest number is the most recently used file.
Exit	Exit the Model Editor.

Edit menu

Use the commands on the Edit menu to edit the graphics and part definition of the current part.

Command	Description

`Cut` 	Cut the selected text.
`Copy` 	Copy the selected text.
`Paste` 	Paste the selected trace after you cut or copy it.
`Delete`	Delete the selected text.
`Find`	Find the specified text. This command is enabled only if the Model Text window is selected.
`Replace`	Invokes the Replace dialog box where you can specify the text to be searched for and replaced. This is enabled only if the Model Text window is selected for simulator models based on device characteristic curves.

View menu

Use the commands on the View menu to change the viewing scale of a plot.

`Command`	Description
`Extract Model` 	Display part definitions in the Spec Entry frame, including graphs. A check beside the command indicates that the item is currently displayed.
`Edit Model`	Display part definitions as editable text. A check beside the command indicates that the item is currently displayed. This is the default view. Is disabled for parameterized or template-based models. For simulator models based on device characteristic curves, part definitions are displayed as editable text. For simulator models with smoke, Test Node Mapping and Smoke parameter window along with part definitions are displayed as editable text in the Model text window.

`Fit` 	Set the display back to the default view factor and redraws the display.
`In` 	Zoom in by a factor of 2 around the point you specify with the mouse.
`Out` 	Zoom out by a factor of 2 around the point you specify with the mouse. The View/Fit limits will not be exceeded. If the View/Fit limits are exceeded in either the X or Y direction, then only the direction in which they are not exceeded will zoom out by the full factor of 2. The exceeded direction will only zoom out to the View/Fit limits.
`Area` 	Cause the display to view into the area you specify with the mouse. You specify this area by holding down the left mouse button and dragging the mouse. This forms a window into which to view. The area may be specified before or after you choose the Area command.
`Previous`	Reset the display to the previous view. Any time the view is changed (with View commands, user-defined axis settings, or with the scroll bars), the view is pushed onto the view stack.
`Redraw`	Redisplay the active plot window.
`Pan-New Center`	Change the center of the plot view without changing the scale of the plot. Click the mouse at the point on the plot that you want to be the center of the new view.
`Toolbars…`	Turn the display of the toolbars on and off.
`Status Bar`	Turn the display of the status bar on and off. A check beside the command indicates that the item is currently displayed.
`Models List`	Turn the display of the Models List frame on and off. A check beside the command indicates that the item is currently displayed.
`Model Text`	Turn the display of the Parameters frame on and off. A check beside the command indicates that the item is currently displayed.

Model menu

Command	Description
New 	Create a new device model definition. See Creating a New Model to learn how to use this command.
Copy From	Copy an existing model definition from a different library to a new name in the current library.
Add Smoke / Undo Add Smoke	Add smoke information to the selected model. The Test Node Mapping and Smoke Parameters forms get added. This command is disabled if you open a template-based simulator model.
DMI Template Code Generator	Generate DMI model code for different PSpice-based DMI models, such as, Analog, Digital, and SystemC. You can also create Verilog-A Compact device models using ADMS in the DMI Template Code Generator. See Generating Template Code for DMI Models for more information on generating the PSpice DMI model template code.
IBIS translator	Translate a model definition in IBIS model format (saved in an .IBS file) to a simulator model definition. The Model Editor supports IBIS version 2.0. **Note:** For IBIS versions later than 2.0, only fields supported for 2.0 will be converted. See IBIS Models to learn how to use this command.
Import	Import model definitions from .MOD files into the current library. See Importing Models to learn how to use this command.
Export	Write the part definitions from the current part to an ASCII text file and give the exported file a .MOD extension, if no extension is specified. Only one model can be exported in a file. Exporting a model to an already existing MOD file overwrites the contents of the MOD file. You can use Export to handle model definitions individually.

Plot menu

Command	Description
Add Trace	Add a trace to the plot. See Displaying Traces to learn how to use this command.
Delete Trace	Remove a trace from the plot. In the device curve plot window, click a trace name to select a trace, then choose Delete Trace to remove the trace from the plot.
Axis Settings	Modify the x-axis and y-axis settings for the selected plot. See Axis Settings to learn how to use this command.

Tools menu

Command	Description
Extract Parameters	Extract model parameters, which means that model parameters are changed according to the specification data. After extraction, all plots are updated to show the new model parameter values.
Customize	Customize toolbars and keyboard shortcuts. See following links to learn how to customize the toolbar and keyboard shortcuts: ■ Customizing Toolbars ■ Customizing Keyboard Shortcuts
Options	Set up automatic part creation for models extracted in the Model Editor. See Setting Model Editor options to learn how to setup automatic part creation. Enable or disable graph splitter synchronization.

Window menu

Command	Description
`Cascade`	Cascade the windows.
`Tile`	Tile the windows.
`Arrange Icons`	Align the minimized windows.
`1`	Bring the corresponding window to the foreground.
`2`	
`3`	
`4`	

Help menu

The Help Menu provides context-sensitive help on all aspects of the Model Editor.

Command	Description
`Help Topics`	Display the topics available in Model Editor Help by viewing Contents, by searching keywords, or by using a full-text search.
`Web Resources`	Access the simulator Community site and the Online help available at this site.
`About Model Editor`	Display the software version, serial number, and OrCAD copyright notice.

Setting Model Editor options

Using the Options dialog box, you can enable spec window splitter synchronization, enable automatic graph updating, view the library search path, change the backup directory, and set up automatic part creation.

To synchronize spec window splitters

When the spec windows are tiled, dragging the split bar of one window drags the split bars of all the other windows in the same manner. You can enable or disable this feature by clicking the Sync Splitters button, or you can select an option in the Options dialog box.

1. From the *Tools* menu, choose *Options*.
2. Under *Misc Settings*, select *Synchronize Graph Splitter Window*.
3. Click *OK*.

To enable automatic graph updating

When this feature is enabled, the graph will be automatically redrawn whenever you add or edit values in the spec. You can enable or disable this feature by clicking the Auto Refresh button, or you can select an option in the Options dialog box.

1. From the *Tools* menu, choose *Options*.
2. Under *Misc Settings*, select *Automatically Update Graph*.
3. Click *OK*.

To view the library search path and change the backup directory

1. From the *Tools* menu, choose *Options*.
2. Under *Misc Settings*, *Current Library Path* displays the order that the Model Editor searches when you edit a model from the design entry tool. You can change the search order in the Simulation Settings dialog box in the design entry tool or simulator.
3. Under *Misc Settings*, in the *Backup* text box, enter the path for the directory where you want the Model Editor to save backup copies of model libraries.
4. Click *OK*.

To set up part creation

The Model Editor can automatically create and configure a new part for each new model you create in the Model Editor.

1. From the *Tools* menu, choose *Options*.
2. Under Part Creation Setup, select the *Always Create Part When Saving Model* check box.

3. To create the symbols in an interactive mode, select the *Pick symbols manually* check box.

   If this check box selected, Model Import Wizard is launched every time a symbol is generated. As a result, you can view the symbols being associated with the simulation model before the .olb file is saved.

4. Select a design entry tool as the schematic editor. All the Cadence schematic editors installed will be listed.

5. Set up the part options you want, as described below.

6. Click *OK*.

Specifying where to save new parts

Under *Save Part To*, specify which part library to save the parts to:

- ❑ Part library path same as model library
- ❑ User-defined part library

Specifying what to base new parts on

Under Base Parts On, enter one of the following in the text box:

- ❑ `MODELED.ETC` to base the symbols on OrCAD's default parts.
- ❑ Another `.OLB` file to base the parts on custom parts from another library.

Part name requirements

The name of the part in a custom part library you specify must match the device type of the model you are creating a part for. See Part file name requirements for information on naming conventions. If the Model Editor cannot find a part with the correct name in the library you specify, the Model Editor uses the corresponding OrCAD default part instead.

Note: For each custom part, set its MODEL property to `` `M `` (where ` is a back single quote). This tells the Model Editor to substitute the correct model name.

Part file name requirements

For this device type…	Use this part name…
Bipolar transistor: lateral PNP	LPNP
Bipolar transistor: NPN	NPN
Bipolar transistor: PNP	PNP
Darlington: N-channel	NDAR
Darlington: P-channel	PDAR
Diode	DIODE
IGBT: N-channel	NIGBT
JFET: N-channel	NJF
JFET: P-channel	PJF
Nonlinear magnetic core	CORE
OPAMP: 5-pin	OPAMP5
OPAMP: 7-pin (with external comp.)	OPAMP7
Power MOSFET: N-channel	NMOS
Power MOSFET: P-channel	PMOS
Voltage comparator	VCOMP
Voltage comparator: 6-pin (with floating ground pin)	VCOMP6
Voltage reference	VREF
Voltage regulator	VREG

Create Parts for Library

Allows you to generate a parts for all the models in a simulator model library.

Enter Input Model Library	Specify the location of the model library for which part symbols have to be generated.

`Browse...`	Use this button to navigate to the model library.
`Enter Output Path`	Specify the location where you want the .OLB to be generated.
`Browse...`	Use this button to navigate to the location where the part library is to be generated.

Setting Up the Page

The standard setup options for printing are offered here along with additional options for determining how model information will be printed. These are found in the Advanced Options frame.

Plots per page

Enter the number of plots you would like to be printed on each page. Specifying only one plot to be printed per page enables the following options to be selected.

- ❑ Include spec data
- ❑ Include active parameter

Enable either of these to have this additional information printed along with your plot.

Parameterized Models supported by the Model Editor

Using the Model Editor, you can create or modify template-based simulator models for the device types listed below.

This device type…	Uses this definition form…	And this name prefix…
Diode	.SUBCKT	X
Bipolar transistor (BJT)	.SUBCKT	X

Ins Gate Bipolar Transistor (IGBT)	.SUBCKT	X
Junction FET (JFET)	.SUBCKT	X
Power MOSFET	.SUBCKT	X
Operational Amplifier	.SUBCKT	X
Magnetic Core	.SUBCKT	K
Voltage Regulator	.SUBCKT	X

Add Smoke

Lists the devices for which the smoke information can be added using the Model Editor.

Using the Model Editor, you can add smoke only to the device types that are supported by the Model Editor. The supported device types are:

- Diode
- Bipolar Junction Transistor (BJT)
- Insulated Gate Bipolar Transistor (IGBT)
- Junction Field Emitter Transistor (JFET)
- Operational Amplifier
- MOSFET (Power MOSFET)
- Voltage Regulator
- Darlington Transistor

The Model Editor fails to recognize the device type of the .SUBCKT models. Therefore, when you use the Model Editor to add smoke information to a device of the .SUBCKT type, you need to specify the device type of the selected model.

Select Physical Parts

The Select Physical Parts dialog box appears when you select a model that has more than one physical device associated to it. These devices have the same simulation model, but different values for the device parameters. For example, the maximum operating limit for the two physical devices having same simulation model can be different.

A list of physical devices attached to the selected simulation model appears in the Parts list box. Select the device for which you want to edit the smoke properties and click OK. Clicking Cancel will not open any model.

Test Node Mapping

Specify the port name that must map to the predefined node listed in the Nodes column. The predefined node names starting with TERM indicate current nodes, and those starting with NODE indicate a voltage node. The default Test Node Mapping information cannot be changed for template-based simulator models. For models based on characteristic curves, users can modify this information.

The information in the Test Node Mapping frame is used for performing the Advanced Analysis smoke tests. The Model Editor can be used to add smoke data for the following device types:

- Diode
- Bipolar Junction Transistor (BJT)
- Insulated Gate Bipolar Transistor (IGBT)
- Junction Field Emitter Transistor (JFET)
- Operational Amplifier
- MOSFET (Power MOSFET)
- Voltage Regulator
- Darlington Transistor

Note: For custom models, such as ones listed above, in the *Test Node Mapping* pane, ensure that the Port names match the Terminal names of the corresponding Capture symbol of the part.

Simulation Parameters

The Simulation Parameters frame appears only when a template-based simulator model is opened in the Model Editor. For each model, the Simulation Parameters frame lists the following:

- Property Name, as it appears in the files used for simulation, such as .lib, device.prp, and device_template.prp.
- Description, of the property names.
- Value, lists the values of the simulation properties to be used in the model. This is an editable field and users can edit the information in this column.
- Default, lists the default values of the simulation properties. Values in this column cannot be edited. During the new model creation, entries in the Value column are same as the entries in the Default column.
- Unit, indicates the unit of measurement for each of the Simulation Parameter.
- Distribution, provides a drop-down list of the distribution functions that are available with Advanced Analysis. This field is enabled only when tolerance values are specified. The distribution functions supported are FLAT, BSIMD.4.2, GAUSS0.4, and SKEW.4.8. By default the distribution function is FLAT.
- Postol, is the positive tolerance for the simulation parameter. If only positive tolerance is specified, the parameter value can range from <value to value+postol>.
- Negtol, is the negative tolerance for the simulation parameter.
- Editable, check box should be selected to ensure that the values specified by the user should override the default values. Selecting this also ensures that the simulation parameter appears in the Model Text and can also be edited from Capture.

Smoke Parameters

The Smoke Parameters frame lists the smoke parameters for the selected model. Smoke parameters are the parameters that describe the maximum operating conditions for a device. The entries in the Smoke Parameters frame are:

- Device Max Ops - This is the name of the maximum operating condition for the device.
- Description - This column provides a small, one-line description for each of the maximum operating condition.

- Value - This is the only editable column in the Smoke Parameters frame. This is the value of the maximum operating condition for the device. This information is obtained from the device data sheet and is entered by the user.
- Unit - This column lists the unit of measurement for each of the maximum operating condition.

For example, consider the following entry in the Smoke Parameters frame,

Device Max Ops	Description	Value	Unit
IPLUS	Max.input current	5	A

This entry indicates that in the selected model, maximum input current allowed is 5 Amperes.

Placing the cursor on each of the parameters in the Smoke Parameters frame displays a tooltip. The tooltip indicates whether the PEAK value, RMS value, or Average value of the selected smoke parameter is used in the smoke analysis.

Reserved words for part creation

During part creation using the Model Editor, you may come across a situation where symbol creation fails for a particular model. This happens when you use one of the reserved words as model name.

Part creation using the Model Editor will fail if you have one of the following as the model name.

AA5_PIN_OPAMP	AA7_PIN_OPAMP	CAP	CORE
AABRDG1	AABRDG3	DARNPN	DARPNP
AABRIDGE	AACORE	DIODE	GASFET
AADARNPN3	AADARPNP3	IND	ISWITCH
AADIODE	AALM117	LPNP	NDAR
AANCHANNEL3	AANIGBT3	NIGBT	NJF
AANIGBT4	AANMOSFET3	NMOS	NPN
AANPN3	AANPN4	OPAMP5	OPAMP7

AAPCHANNEL3	AAPMOSFET3	PDAR	PJF
AAPNP3	AASCR	PMOS	PNP
AATRANSIENT_SUPPRESSOR	AAVREG	RES	TRN
AAZENER_DIODE	VREG	TRN1	VCOMP
VSWITCH	VREF	VCOMP6	ZENER

There are some exceptions to this rule. For example, part creation using the Model Editor will be successful only if you use aa5_pin_opamp as the model name for a template-based five-pin opamp model. For all other cases the part creation process will fail. In spite of the exceptions, it is recommended that the reserved words should not be used as model names.

Enabling auto part creation

To enable auto part creation in the Model Editor:

1. Choose *Options* from the *File* menu.
2. In the Options dialog box, select the *Always Create Part When Saving Model* check box.

Now every time you save a model, the symbol for the same will be updated. The location of the part library generated, depends on the selection made by you in the Save part to group box. For details see, Setting Model Editor options.

To enable the users to view the symbol shapes before these are associated with a model, select the Pick symbols manually check box. Selecting the check box ensures that Model Import Wizard is launched, to obtain user inputs, every time a symbol is generated.

Creating parts in batch mode

Using the Model Editor, you create design entry tool symbols for all the models in a library.

1. From the File menu choose *Export to <design entry tool> Part Library*.
2. Specify the location of the model library for which you need to create design entry tool symbols.
3. Specify the location where you want the part library to be generated.
4. Click *OK*.

5. Close the message box that appears after the part creation process is complete.

Creating parts in an interactive mode

Model Import Wizard allows you to create design entry tool parts in an interactive mode. To use Model Import Wizard:

1. From the File menu, choose *Model Import Wizard [<design entry tool>]*.
2. Specify the location of the model library for which you need to create design entry tool symbols.
3. Specify the location where you want the part library to be generated.
4. To start the process of creating parts, click *Next*.

   In the Associate/Replace Symbol page, you can preview the symbols associated with each of the model in the library.
5. To attach symbols to the models, for which the Model Import Wizard could not find matching symbols, select the model and click the *Associate Symbol* button.

   To replace symbols associated by the Model Import Wizard, select the model and click the *Replace Symbol* button.
6. Specify the symbol library that contains the required symbol.

   Model Import Wizard generates a list of matching symbols that can be associated with the selected model.
7. Select a symbol and click *Next*.
8. Specify the symbol pin to be associated with each model terminal.
9. To save your changes, click the *Save Symbol* button.
10. Similarly, you can associate or replace symbols for all the models in the library. To complete the process, click the *Finish* button.

    All the changes are saved in the destination symbol library.

Associate/Replace Symbol Page

Models with symbol	Select this check box, to list the models, for which the Model Import Wizard can find matching symbols, along with the corresponding symbol names.
Models without symbol	Select this check box, to list the models for which matching symbols could not be found.
Associate/Replace Symbols	Select this toggle button when you want to attach an existing symbol to the selected simulation model.Associate Symbol button is available when you select a model from the Models without symbols list. Use the Associate symbol button to associate an existing symbol to a model without symbol. Replace Symbol button is available when the selected model already has a symbol associated to it. Use the Replace symbol button to replace a symbol associated to a model, by an existing symbol of your choice.
Back	This button is available only when the Model Import wizard is launched using the Model Import Wizard command from the File drop-down menu.Use this button to move back to the Specify Library page of the wizard.
Finish	Select this if you want to stop the process of creating symbols.In case you have models that do not have any symbols associated to them, a message box appears, asking you whether rectangular symbols should be attached to these models or not.
`View Model Text`	Displays the model text for the selected model in a new window.

Select Matching Page

Select a library containing matching symbols	Specify the library that contains the symbol you want to associate the selected model.You can either use the browse button to navigate to the desired olb file, or select the file from the drop-down list box. The drop-down menu lists a maximum of 10 most recently used libraries.
Matching Symbols	Lists all the symbols that can be associated with the model selected in Associate/Replace Symbol page. The matching symbols list is generated based on model definition of the selected model by the user.

Symbol pane	Displays the graphical shape of the symbol currently selected in the Matching Symbols list.
Next	Select this to move to the next step.
Cancel	Select this to cancel the process of associating an existing symbol to a simulation model.
View Model Text	Displays the model text for the selected model in a new window.

Define Pin Mapping Page

Use this page for pin to port mapping between the selected symbol shape and the model definition.

While you complete the pin-port mapping, you can view the symbol shape in the Symbol pane on the right of the wizard, and the use the View Model Text button to view the model definition.

Note: All the symbol pins must be mapped to a model terminal. After you have mapped each symbol pin to a unique model terminal, if there are any optional model terminals left, you may leave them unmapped.

`Model Terminal`	Lists the port names from the model definition
`Symbol Pin`	Lists the symbol pin names.From the drop-down list, select the pin name that is to be associated with the listed model terminal.
`Optional Model Terminals`	Lists the optional ports in the model definition.Depending on the availability of symbol pins, you may or may not map these.
`Symbol window`	Displays the shape of the symbol selected in the Matching Symbols list.
`Back`	Select this to move to the previous step, where you selected a matching symbol.
`Cancel`	Select this to cancel the process of associating an existing symbol to a simulation model.
`Save Symbol`	Select this to complete the process of associating a symbol to the selected model and to jump back to the Associate/Replace symbol page.
`View Model Text`	Displays the model text for the selected model in a new window.

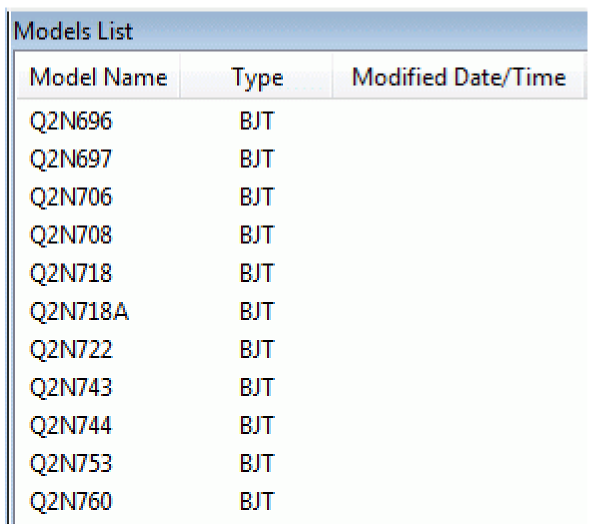
Specify Library Page

Enter Input Model Library	Specify the name and the location of the .lib file containing the models for which symbols are required.
Enter Destination Symbol Library	Specify the name and the location of the symbol library (.olb) to be created.

Performing Model Editor Tasks

Using the Models List Window

The Models List window, located on the left side of the workspace, displays the names of the models contained in the current library (if any), as well as each model's device type and creation date. You can dock this window to any side of the workspace.



You can sort the models by clicking the appropriate header. If a type is unknown, the entry in the column is UNKNOWN. If a creation date is unknown, the entry in the column is blank.

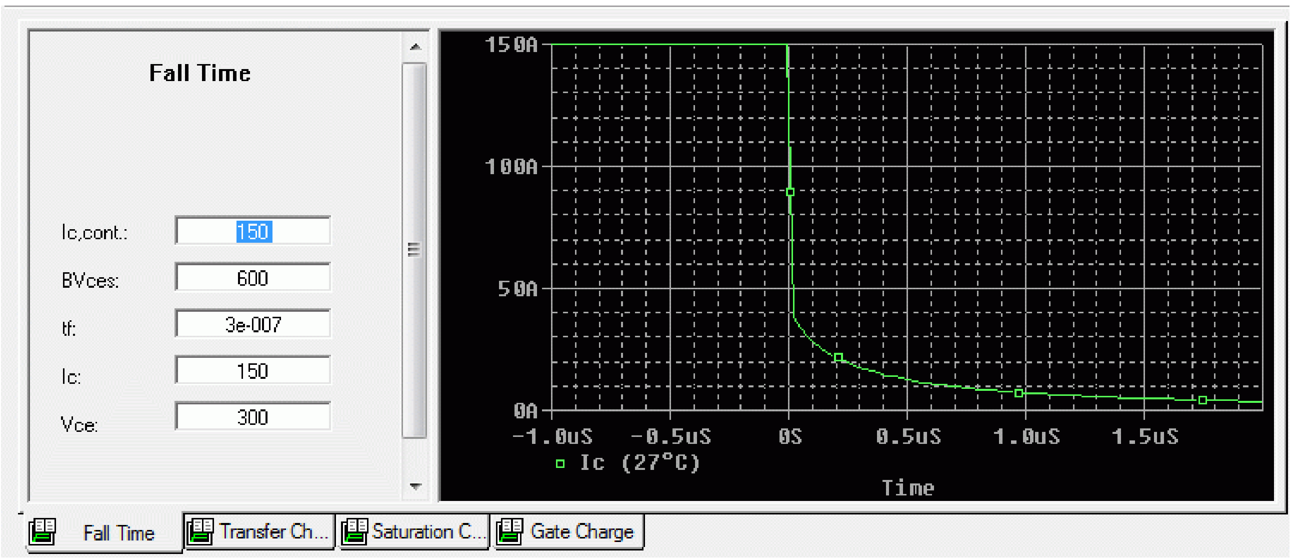
VHDL models are unknown

You can display a model for editing by selecting the model name. Depending on the type of the model selected, the model information is displayed in different windows. If the model is a template-based simulator model, the simulation parameters frame displays along with the Model Text window in the read-only mode.

Selecting a simulator model based on device characteristic curves and with smoke information, displays the Test Node mapping and Smoke Parameters frame. When you select a model with out the smoke information either the model's graph is displayed in the Spec Entry window for the models supported by the Model Editor. For more information, see Getting Models from a Library. If the model is not recognized by the Model Editor or is a subcircuit definition, the model text is displayed instead of the tabbed windows. See Editing Model Text for more information.

Using the Spec Entry Frame

When you choose a new or existing model for editing (from the Models List window), a series of maximized windows is opened in the Spec Entry window, one for each spec.



Using the tabs

The spec windows correspond to tabs shown at the bottom of the Spec Entry window. The top spec window corresponds to the left tab, and the left-to-right order of the tabs is the recommended order for entering and extracting parameters. You can navigate among these spec windows by clicking the tabs.

Entering device data

Each spec window contains two frames: one on the left that has text boxes for entering data, one on the right that displays a relevant graph. Pressing F1 displays Help for the active spec window.

Using the graphs

A spec's graph shows any data points entered by the user and the calculated graph. If a spec has no graph, the graph area contains a statement that there is no graph for that spec. Update the graph after entering new data by clicking the Update Graph button.

Note: Some graphs do not use table input data, so no points are displayed.

Arranging the windows

The spec windows are separated by split bars so you can enlarge, reduce, or conceal the viewing area of each.

You can also be cascade or tile these spec windows (using the Window menu) so that you can view all spec data at once and compare their graphs. When a window is minimized, maximize it by double-clicking its title bar.

When the spec windows are tiled, dragging the split bar of one window drags the split bars of all the other windows in the same manner. You can enable or disable this feature by clicking the Sync Splitters button or by selecting an option in the Options dialog box.

Using the Parameter Window

Use the Parameter window to edit model parameters directly and investigate how changing parameter values affects a device characteristic. You can dock this window to any side of the workspace.

Parameters

Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
TAU	7.1e-006	1e-009	0.0001	7.1e-006	☑	☐
KP	0.38	0.12	100	0.38	☐	☐
AREA	1e-005	1e-012	0.001	1e-005	☑	☐
AGD	5e-006	1e-007	0.001	5e-006	☑	☐
WB	9e-005	1e-009	0.001	9e-005	☑	☐
VT	2	0.8	100	2	☐	☐
MUN	1500	0.0001	1000000	1500	☐	☐
MUP	450	0.0001	1000000	450	☐	☐
BVF	1	0.0001	10	1	☐	☐
NB	2e+014	1	1e+030	2e+014	☐	☐

As you change model parameters, the Model Editor recalculates the behavior of the model characteristics and displays a new curve for each of the affected ones in the Spec Entry window.

For information on how to edit Parameter values, see To edit model parameters.

Using the columns

The Parameter Name column is the list of parameters for the current device type.

The Value column displays the current value of each parameter. The Default column displays the default value of the parameter.

The Minimum and Maximum columns display the upper and lower boundaries for the parameter during extraction.

The Active column displays which parameters apply to the device characteristic in the active Spec Entry window.

The Fixed column displays the fixed-value status of each parameter. Select the fixed check box in a parameter row to keep the current value as specified during parameter extraction.

Tip

You can sort the rows by clicking any header buttons; click once to sort in descending order and click again to reverse the order. In the Value, Minimum, and Maximum columns, each cell is a text box you can edit.

Getting Models from a Library

To get a model from a library:

1. Choose *File – Open* and then select a model library to open and click *OK*.

   The model names in the library are displayed in the Models list window, located on the left side of the workspace.

2. In the Models List window, select a model name to display the model for editing. See Editing Model Specs and Parameters for more information on editing models.

3. Click *Save* to save changes to the model and library, or from the File menu, choose Save As to save the model and library under a different name. See Saving Global Models (and Parts) for more information.

Note: The Model Editor parameter defaults are different from the simulator default parameters. Refer to the Analog Devices chapter of the online simulator Reference Manual for the default simulator parameters.

Importing Models

Note: You must have a library open to import a model into.

To import a model

1. Choose *Model – Import*.
2. Select the .MOD file that contains the model you want to import, then click *OK*.

Note: Template-based simulator models are imported without the device information in the .PRP file.

Generating Template Code for DMI Models

You can generate the following type of DMI models from Model Editor:

- Analog
- Digital C/C++
- SystemC
- VerilogA-ADMS

Important

Ensure the following before using the DMI Template Code Generator feature:

- You have one of the following licenses:
  - *OrCAD PSpice Designer Plus*
  - *Allegro PSpice Simulator*
  - *PSpice Systems Option*
- You have installed Microsoft Visual Studio Community 2013 on your machine to generate PSpice DMI Dynamic Link Library (.DLL) files.
- For VerilogA-ADMS DMI Models, you can also use Microsoft Visual Studio Express 2012 to generate PSpice DMI DLL files, but you need to update `"%VS120COMNTOOLS%"` to `"%VS110COMNTOOLS%"` in the `buildADMS_Tcl.bat` batch script located at the following path: `<Installation_Diretctory>\tools\pspice\tclscripts\pspModelCreate\VerilogATemplate`

Analog DMI Models

By selecting the Analog option as Part Type in the DMI Template Code Generator window, you can following type of the models:

- Generic Device
- Voltage-Controlled Voltage Source
- Function-Dependent Voltage Source
- Voltage-Controlled Current Source
- Function-Dependent Current Source

- Generic Two-Node Device
- Generic Three-Node Device

You can specify different parameters that are required for the models, such as Global Parameters, Device Parameters, and Model Parameters. A default model parameter, `MaxStepSize`, is predefined in the dialog box for all the model types.

Note: You can specify the Terminal points for the generic device only. For other devices, the number of terminal points are non-editable, but you can edit the name and description of terminal points.

Once you generate the .lib file using the DMI Template Code Generator, the following files will be generated along the .lib:

- `<ModelName>.lib` - The PSpice Model (.lib) generated using DMI Template Code Generator
- `<ModelName>.log` - The log file that tells if code generation is success or not
- `<ModelName>_user.cpp` - The C++ source file that contains logic related to the DMI model code
- `psp<ModelName>.cpp` - The C++ source file that contains logic related to the DMI model code
- `psp<ModelName>.h` - The C++ header file that contains logic related to the DMI model code
- `pspEngFunc.cpp` - The mandatory C++ source file to generate PSpice library file
- `pspEngFunc.h` - The mandatory C++ header file to generate PSpice library file
- `PSpiceBase.h` - The mandatory C++ header file to generate PSpice library file
- `PSpiceCMIApiDefs.h` - The mandatory C++ header file to generate PSpice library file
- `PSpiceCommonAPIDefs.h` - The mandatory C++ header file to generate PSpice library file
- `PSpiceDigApiDefs.h` - The mandatory C++ header file to generate PSpice library file
- `StdAfx.cpp` - The mandatory C++ source file to generate PSpice library file
- `StdAfx.h` - The mandatory C++ header file to generate PSpice library file

You can edit `<ModelName>_user.cpp`, `psp<ModelName>.cpp`, and `psp<ModelName>.h` in Microsoft Visual Studio Express 2013.

Digital C/C++ DMI Models

To generate a Digital C/C++-based DMI model using the DMI Template Code Generator dialog, select `Digital C/C++` as Part Type.

The dialog box provides you an option specify the interface type of the Digital C/C++ model, such as `Combinatorial` and `Clocked`. You can also specify Input Ports and IO Ports using a CSV file.

Note: In case of a clocked interface, a default CLK port is predefined if the ports are specified using the DMI Template Code Generator dialog.

Once you generate the .lib file using the DMI Template Code Generator, the following files will be generated along the .lib:

- `<ModelName>.lib` - The PSpice Model (.lib) generated using DMI Template Code Generator
- `<ModelName>.log` - The log file that tells if code generation is success or not
- `<ModelName>_user.cpp` - The C++ source file that contains logic related to the DMI model code
- `psp<ModelName>.cpp` - The C++ source file that contains logic related to the DMI model code
- `psp<ModelName>.h` - The C++ header file that contains logic related to the DMI model code
- `pspEngFunc.cpp` - An auto-generated C++ source file related to PSpice Engine tasks
- `pspEngFunc.h` - An auto-generated C++ header file related to PSpice Engine tasks
- `PSpiceBase.h` - A mandatory C++ header file
- `PSpiceCMIApiDefs.h` - A mandatory C++ header file
- `PSpiceCommonAPIDefs.h` - A mandatory C++ header file
- `PSpiceDigApiDefs.h` - A mandatory C++ header file
- `StdAfx.cpp` - A mandatory C++ source file
- `StdAfx.h` - A mandatory C++ header file

You can edit `<ModelName>_user.cpp`, `psp<ModelName>.cpp`, and `psp<ModelName>.h` in Microsoft Visual Studio Express 2013.

Note: If you generate a new .lib file in a folder that contains an already generated .lib file with other required files, only the following files will be newly generated: `<NewModelName>_user.cpp`, `psp<NewModelName>.cpp`, and `psp<NewModelName>.h`

SystemC DMI Models

Select `SystemC` as Part Type in the DMI Template Code Generator dialog to generate SystemC-based PSpice DMI models. You can specify Input Ports and IO Ports using a CSV file or from the dialog box directly.

Only Clocked type interfaces are supported for SystemC-based PSpice DMI models. A default CLK port is predefined if the ports are specified using the DMI Template Code Generator dialog

Once you generate the .lib file using the DMI Template Code Generator, the following files will be generated along the .lib:

- `<ModelName>.lib` - The PSpice Model (.lib) generated using DMI Template Code Generator
- `<ModelName>.log` - The log file that tells if code generation is success or not
- `pspSysC<ModelName>_user.cpp` - The C++ source file that contains logic related to the DMI model code
- `pspSysC<ModelName>.cpp` - The C++ source file that contains logic related to the DMI model code
- `pspSysC<ModelName>.h` - The C++ header file that contains logic related to the DMI model code
- `pspEngFunc.cpp` - An auto-generated C++ source file related to PSpice Engine tasks
- `pspEngFunc.h` - An auto-generated C++ header file related to PSpice Engine tasks
- `PSpiceBase.h` - A mandatory C++ header file
- `PSpiceCMIApiDefs.h` - A mandatory C++ header file
- `PSpiceCommonAPIDefs.h` - A mandatory C++ header file
- `PSpiceDigApiDefs.h` - A mandatory C++ header file
- `StdAfx.cpp` - A mandatory C++ source file
- `StdAfx.h` - A mandatory C++ header file

- SysC<ModelName>.cpp - A mandatory C++ source file for SystemC DMI model
- SysC<ModelName>.h - A mandatory C++ header file for SystemC DMI model

You can edit <ModelName>_user.cpp, psp<ModelName>.cpp, and psp<ModelName>.h in Microsoft Visual Studio Express 2013.

VerilogA-ADMS DMI Models

Using the DMI Template Code Generator dialog, you can generate a PSpice DMI model from the VerilogA-based logic. The generated DMI model can used in Capture-PSpice flow.

To generate a PSpice DMI model from a VerilogA logic, select VerilogA-ADMS as Part Type in the dialog box. The default path for XML template files is <Installation>/tools/pspice/api/adms/xmls.

Note: The Part Name should be same as the VerilogA filename.

Limitations of VerilogA Modeling

VerilogA modeling has the following limitations:

- The ddt function is supported in the contribution statements only.
- The idt function is not supported.

  Alternatively, while modeling a capacitor, differentiate the charge d/dt (q) and contribute it to a current source.
- The equations where potential is on the left side and current on the right side, such as V <+ I cannot be processed.

  Note: The following equation cannot be processed: V(a,b) <+ ddt(L*I(a,b)) cannot be processed. To process this equation, add a dummy command, I(a,b) <+ (0)*(ddt(V(a,b)) before the given command. The dummy command does not impact the operation as it multiplies the expression with 0.

IBIS Models

This section covers the introduction and translation of the IBIS models.

Overview

I/O Buffer Information Specification known as IBIS is used to behaviorally represent the Input, Output, and I/O buffers of digital ICs. The IBIS models are used to perform signal integrity (SI) simulations and timing analysis of printed circuit boards (PCBs). These models characterize a buffer using a collection of DC data (current vs. voltage), as well as transient data (voltage vs. time) in form of I-V and V-t tables. The IBIS models are used because they are easily available from IC vendors and they do not disclose proprietary information about circuits and processes.

PSpice users perform SPICE simulation on the complete design, which includes digital ICs whose pin buffer models are available in the IBIS format. To achieve this, the IBIS model needs to be converted to the PSpice model. The translated PSpice model is a subcircuit that converts the V-I and V-t data from IBIS file to PSpice compatible tables.

The operating supply voltage present in the IBIS file in form of `Voltage Range` or `Pullup`, `Pulldown`, `POWER`, and `GND` references is also read and interpreted in the PSpice subcircuit. The resulting PSpice subcircuit does not have any power pins because they are already included in the subcircuit.

Translating IBIS Models

You can invoke the IBIS translator either from the Windows command prompt using the `orPSpiceParsers.exe` utility or from the Model Editor. The supported files are `.ibs` and `.dml`.

The IBIS translator supports all versions supported by Signoise but translates only supported keyword.

- Translating IBIS Models from the Command Prompt
- Translating IBIS Models Using Model Editor
- Supported Keywords

Note: Algorithmic Modeling Interface (AMI) sections of IBIS models are not supported by the IBIS to PSpice translator.

Translating IBIS Models from the Command Prompt

To translate IBIS models from the command prompt, enter the following command:

```
orPSpiceParsers.exe [parameter1,[parameter2]..] <IBIS File Name>|<DML File Name>
```

Where

- *<IBIS File Name>*: Is the IBIS file that contains IBIS model(s).
- *<DML File Name>*: Is the name of the DML file. Device Modeling Language (DML) is a another format used to represent multiple IBIS models. The DML format is used in the Allegro PCB Signal Integrity (SI) flow.
- `parameter`: The following table lists the various optional parameters:

`-model <Model Name>`	Generates a sub-circuit for only the specified model in the output lib
`-type <typ|max|min>`	Generates model of the specified type, namely: `typ` (Typical), `max` (Maximum), `min` (Minimum), or `all`. The default is `typ`.
`-RFix <value> -VFix <value>`	Specifies the V-t table to be used for generating the output characteristics of the model. The default is to use V-t table with minimum `RFix`.
`-Use2Vt`	Uses two Rising/Falling waveform tables. By default, only one table is used.
`-ROSNB <value>`	Specifies the value of snub resistance to be added across `L_pkg`. By default `ROSNB = 20 ohms`.
`-stim`	Specifies that a stimulus is created automatically for the output and IO buffers.
`-pinModels`	Specifies that wrapper models for all signals defined in the IBIS file are also created. By default, PSpice macro-models are created only for the buffer models defined in the IBIS file.

`-skipDmlCheck`	Specifies that the DML check is skipped and output library is generated irrespective of any warnings or errors present in the DML file. **Note:** By default, this option is ON when IBIS models are translated using Model Editor.

This command creates the following files:

- A library file with the name `<Input_File_Name>.lib`. This file contains PSpice macro-models for all Buffer Models defined in the IBIS file. It might also contain wrapper models for all signals defined in the IBIS file.

  The models corresponding to IBIS buffer models are named `<File_Name>_<Component_Name>_[Min|Max|Typ]` and the models corresponding to IBIS signals are named `<Signal_Name>_[Min|Max|Typ]`.

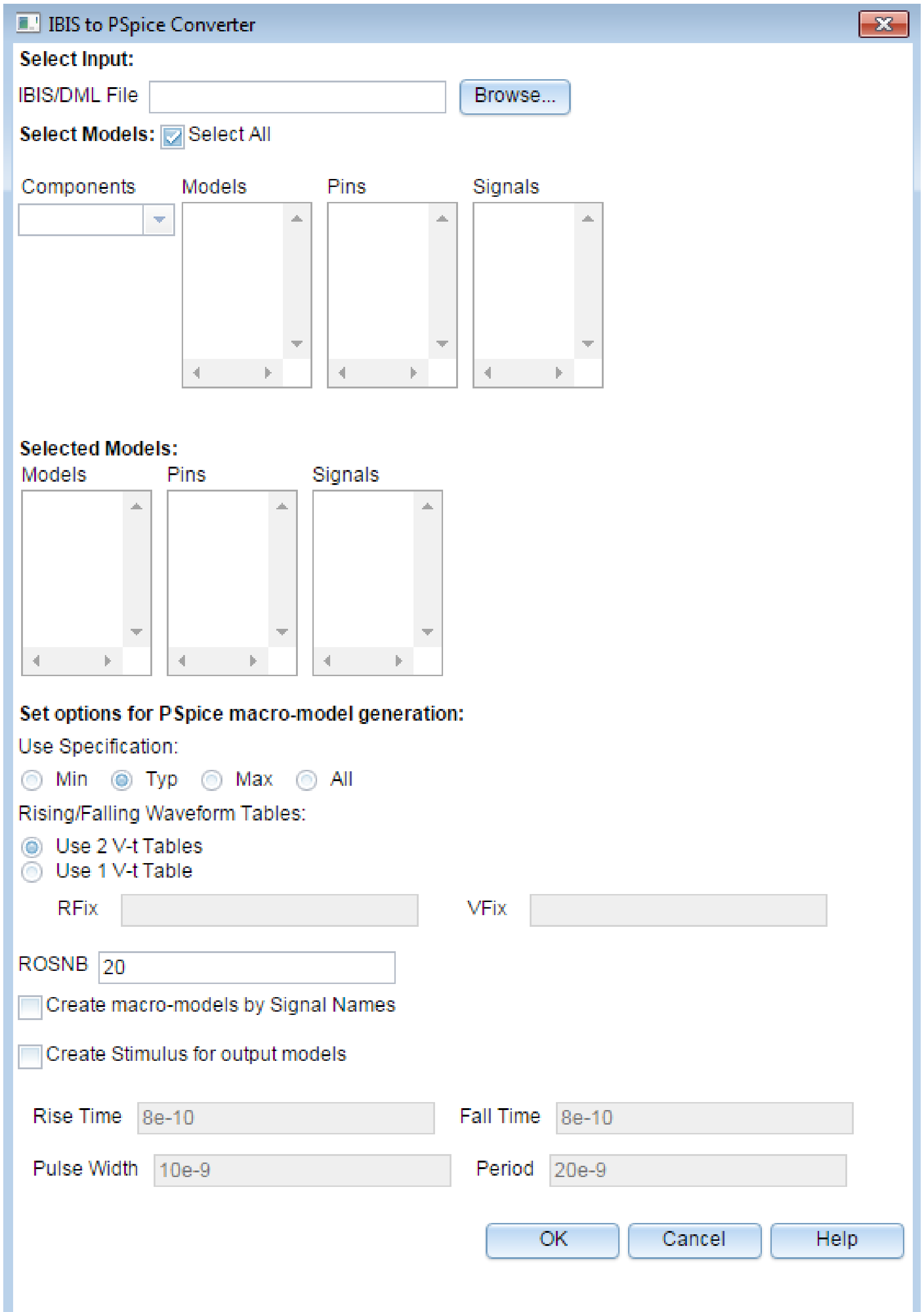
- A log file with the name `<Input_File_Name>_ibis2pspice.log`.

Translating IBIS Models Using Model Editor

To translate an IBIS model using Model Editor:

1. Open Model Editor.
2. Choose *Model – IBIS Translator*.

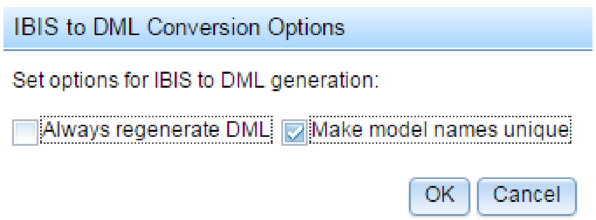
The IBIS translator appears.



Note: If you select the *Create Stimulus for output models* option, the system generates a stimulus internally for output models and the output models generated have 2 ports, that are; OUTPUT, ENABLE. If this option is unselected, the output models generated have 3 ports with user specifying input from outside. The input stimulus generated is of PULSE type, with voltage levels as specified in IBIS model. Other parameters such as, *Pulse Width*, *Period*, *Rise Time*, and *Fall Time* are specified using the IBIS translator GUI.

3. In the *IBIS/DML File* field, specify a `.ibs` or `.dml` file to be translated in the IBIS translator dialog box. You can use the *Browse* button to select the file.

   A dialog box appears for IBIS to DML generation settings.



   Select:

   - *Always regenerate DML to* regenerate DML files. IBIS files are first converted to their equivalent dml files, and the dml files are then read. If *Always regenerate DML* is not selected, IBIS to DML conversion is not done if a dml file with the same name is already available and the dml is more recent that the IBIS file.
   - *Make model names unique* to create models with unique names by prepending the file name to the model name.

4. Click *OK* to close the dialog box.
5. In the IBIS translator dialog box, select the *Select All* option if you want to select all models. To select individual models, deselect *Select All* and then select the listed models.
6. Select the *Use Specification* option to select the type of models you want to generate: `Typ` (Typical), `Max` (Maximum), `Min` (Minimum), or `All`. The default is `Typ`.
7. Select either *Use 2 V-t Tables* (default) to use two rising/falling waveform tables or *Use 1 V-t Table* to use a single waveform table.

   If you select *Use 1 V-t Table*, you can also specify the *RFix* and *VFix* values for the table that you want to be used for generating the PSpice model. By default, the V-t table with the lowest RFix will be used.

8. Click *OK*.

   This creates the following files:

- A library file with the name <Input_File_Name>.lib. This file contains PSpice macro-models for all buffer models defined in the IBIS file. It might also contain wrapper models for all Signals defined in the IBIS file.

  The models corresponding to IBIS buffer models are named <File_Name>_<Component_Name>_[Min|Max|Typ] and the models corresponding to IBIS signals are named <Signal_Name>_[Min|Max|Typ].

- A log file with the name <Input_File_Name>_ibis2pspice.log.

Supported Keywords

Irrespective of the IBIS version, the translator supports only the following keywords:

[Package]	
	R_pkg
	L_pkg
	C_pkg
[Pin]	
	signal_name
	model_name
	R_pin
	L_pin
	C_pin
[Model]	
	Model_type
	Polarity, Enable
	Vinl
	Vinh
	C_Comp
[Pulldown]	
[Pullup]	
[GND Clamp]	

```
[POWER Clamp]
[Rising Waveform]
                    R_Fixture
                    V_Fixture
[Falling Waveform]
                    R_Fixture
                    V_Fixture
[Ramp]
```

Editing Model Specs and Parameters

Load a model for editing by creating a new model, getting an existing model, or importing a model from a library.

To edit a model specification

1. In the Models List window, select the model whose spec you want to edit.
   - ❑ If the model is recognized by the Model Editor, specifications are displayed in the Spec Entry window.
   - ❑ If the model is unrecognized by the Model Editor or is a subcircuit definition, the model text is displayed instead of the tabbed windows. See Editing Model Text text for more information.
2. In the Spec Entry window, click the tab for the characteristic you want to edit.
3. Add, modify, and/or delete specification data as needed.
   - ❑ Click the Update Graph button to view changes to the device graph.
   - ❑ Pressing F1 displays Help for the current tab.
4. Click the Extract Model Parameters button to update model to include the new characteristics.
5. Click Save to save changes to the model and library, or from the File menu, choose Save As to save the model and library under a different name. See Saving Global Models (and Parts) for more information.

To edit model parameters

1. In the Parameters window, scroll the list of parameters and find the one you want to edit.

   The Active check box is selected for all the parameters that apply to the active characteristic in the Spec Entry window.

2. In the Value text box, type a new value.

   The Default text box displays the default value for the parameter.

3. In the Minimum and Maximum text boxes, specify upper and lower boundaries for the parameters extracted.

   Select the Fixed check box to keep the current value as specified during extraction.

4. Click Save to save changes to the model and library, or from the File menu, choose Save As to save the model and library under a different name. See Saving Global Models (and Parts) for more information.

Editing Model Text

Model Text is editable only for simulator models that are based on device characteristic curves. For template-based models the Model Text appears in the read-only mode.

Load a model for editing by creating a new model, getting an existing model, or importing a model from a library.

1. In the Parts List window, select a model. If the selected model is:

   - A device characteristic curve-based model created using the Model Editor, the model specifications are displayed in the Spec Entry window. To display the model text instead of the tabbed windows, from the View menu, choose Edit Model.

   - A device characteristic curve-based model from a Cadence supplied library, the Test Node Mapping window, the Smoke Parameters window, and the Model Text window appear. In this case you can edit the contents of the Model Text window. To view the Spec Entry window and the Parameters window, choose Extract Model from the View menu.

   - A template-based/parameterized model, the Simulation Parameters window and the Model Text window appear. In this case, the Model Text window is not editable. The model text appears in the read-only mode. The Extract Model and the Edit Model commands in the View menu are also disabled.

2. Edit the model text as needed. See the online simulator Reference Manual for descriptions of each device type.
3. From the File Menu, choose Save to save the library. The model information is also saved. See Saving Global Models (and Parts) for more information.

Note: The Model Editor parameter defaults are different from the simulator default parameters. Refer to the Analog Devices chapter of the online simulator Reference Manual for the default simulator parameters.

Copying Models

To copy a model

1. Open an existing library or create a new library.
2. From the Model menu, choose Copy From to display the Copy Model dialog box.
3. In the New Model text box, type a unique name for the new model.
4. In the From Model list, select the name of the model to copy.

   The current library name is displayed in the Source Library text box. To select a model from another library, click Browse and select another library.
5. Click OK to copy the model definition to the new model name.

Displaying Traces

To display one trace

1. In the Spec Entry window, click a tab.
2. If the corresponding trace is not displayed, click the Update Graph button.

To display multiple traces

1. In the Spec Entry window, click a tab.
2. From the Plot menu, choose Add Trace.
3. Specify the variable value for the new trace and click OK.

Creating a New Model

You must have a library open to create a new model.

To create a new model

1. From the Model menu, choose New, or click the New Model button {bmc newpart.bmp} to display the New dialog box.
2. In the Model text box, type a unique name for the new model.

   The model name can be alphanumeric. The only special characters that can be included in the model name are $, _, *, /, %, -, and +.
3. To create a regular simulator model, select the Use Device Characteristic Curves radio button.

   or

   To create a parameterized model, select the Use Templates radio button.
4. In the From Model list box, enter a model type. Choose any other model type information needed (such as input polarity), then click OK.
5. Default specifications are displayed in the Spec Entry frame, located on the right side of the workspace. You can edit the model specifications or parameters.
6. From the File Menu, choose Save to save the library and its models. The model information is also saved. See Saving Global Models (and Parts) for more information.

Note: The Model Editor default parameters are different from the simulator default parameters. Refer to the Analog Devices chapter of the online simulator Reference Manual for the default simulator parameters.

Saving Global Models (and Parts)

When you have finished editing models, the Model Editor does the following for you:

- Saves the model definition to the open library.
- If you had the automatic part creation option enabled, saves the part definition to the .OLB file you specified.
- If the model and/or part library is new, configures each for global use.

To save the new model (and part)

Choose File-Save to update the library and save the MODEL_LIBRARY_NAME.LIB file (and, if you enabled part creation, the.OLB file you specified).

To save the open model library as a new library

1. Choose File - Save As.
2. Enter the name of the new model library you want to save the definition to.

   The Model Editor still automatically configures the model library as global. If the Model Editor created a part, the Model Editor saves the part definition to the .OLB file you specified, which is also global.

To export only the model definition to a different library

1. From the Model menu, choose Export.
2. Enter the name of the new library.

Customizing Toolbars

Toolbar settings can be saved in settings schemes.

- To change the display of toolbars
- To create a new toolbar
- To add buttons to toolbars
- To remove buttons from toolbars
- To reset toolbars to their default settings

To change the display of toolbars

1. Choose Tools-Customize.
2. Click the Toolbars tab.
3. In the Toolbars list, select (?) the toolbars you want to be displayed.
4. Select any of the following options:

- ❑ Select (?) Show Tooltips to enable Tooltips.
- ❑ Select (?) Cool Look to make the toolbar buttons appear flat.
- ❑ Select (?) Large Buttons to display the toolbar buttons at a larger size.

5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

To create a new toolbar

1. From the Tools menu, choose Customize.
2. Click the Toolbars tab.
3. Click New.
4. In the text box that appears, type a name for the toolbar, then click OK.

   The name of the new toolbar appears in the Toolbars list.
5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

Tip

You can also create a new toolbar by dragging a button from the Customize dialog box to any open area on the workspace.

To add buttons to toolbars

1. Choose Tools-Customize.
2. Click the Commands tab.
3. In the Categories list, click a category to display related toolbar buttons in the Buttons frame.
4. In the Buttons frame, click a button to display a description of its function in the Description frame.
5. To add the selected button to a toolbar, drag it from the Customize dialog box to any toolbar displayed in the program window.
6. Click OK to close the dialog box.

Note: Changes are applied even if you don't click Apply.

To remove buttons from toolbars

1. Drag the button you do not want from the toolbar to the Customize dialog box.

To reset toolbars to their default settings

1. Choose Tools-Customize.
2. Click the Toolbars tab.
3. Do one of the following:
   - ❑ Under Scheme, from the list, select Default.
   - ❑ Click Reset.
4. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

Note: Changes are applied only to the currently selected toolbar.

Customizing Keyboard Shortcuts

Keyboard shortcuts can be saved in settings schemes.

- To create a keyboard shortcut
- To remove a keyboard shortcut
- To reset all keyboard shortcuts to their default settings

To create a keyboard shortcut

1. From the Tools menu, choose Customize.
2. Click the Keyboard tab.
3. In the Select a Command frame, select a command to display its description in the Description frame.

   Any shortcuts already assigned to the selected command are displayed in the Assigned Shortcuts frame.
4. Click Create Shortcut.

5. In the dialog box that appears, enter the shortcut that you want to assign to the command, then click OK.

6. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

To remove a keyboard shortcut

1. From the Tools menu, choose Customize.

2. Click the Keyboard tab.

3. In the Select a Command frame, select the command whose shortcut you want to remove.

   A description of the command is displayed in the Description frame. Any shortcuts already assigned to the selected command are displayed in the Assigned Shortcuts frame.

4. In the Assigned Shortcuts frame, select the shortcut you want to remove, then click Remove.

5. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

To reset all keyboard shortcuts to their default settings

1. From the Tools menu, choose Customize.

2. Click the Keyboard tab.

3. Do one of the following:

   ❑ Under Scheme, from the list, select Default.

   ❑ Click Reset.

4. Click Apply to apply changes without closing the dialog box, or click OK to apply changes and close the dialog box.

Settings Schemes

Toolbar and keyboard shortcut settings can be saved in schemes. This is useful for people sharing a computer; they can each save their settings to suit their own work habits.

- To apply a scheme
- To create a scheme
- To delete a scheme

To apply a scheme

1. Choose *Tools – Customize*.
2. Under Scheme, from the list, select the name of the scheme you want to use.
3. Click *Apply* to apply changes without closing the dialog box, or click *OK* to apply changes and close the dialog box.

To create a scheme

1. From the Tools menu, choose Customize.
2. Set up your toolbar and keyboard shortcut preferences, while clicking Apply to keep the dialog box open.
3. Under Scheme, click Save As.
4. In the text box that appears, type a name for your scheme, then click OK.
5. Click OK to close the dialog box.

To delete a scheme

1. From the Tools menu, choose Customize.
2. Click any of the tabs.
3. Under Scheme, select the name of the scheme you want to delete, then click Delete.
4. When asked whether you are sure you want to delete the scheme, click Yes.
5. Click OK to close the dialog box.

Axis Settings

`Data Range`	Auto Range sets the range of the axis to be the range of its variable/traces, rounded to a convenient value. The range of the axis will be automatically adjusted as its variable/traces are changed. User Defined sets the range of the axis to your specification. Once set, the range is not affected by changes in the axis' variable/traces. It can only be changed by one of the view commands or by setting another User Defined range. To enter a data range, enter the beginning value of the range in the first field, then enter the end value in the second field. The units you specify will appear on the numbers labeling the axis' tick marks. This command may be used to reverse an axis. For instance, instead of having a range of (0V to 5V), you can set to range (5V to 0V). All the traces on that axis are then reversed.
`Scale`	Linear sets the current axis to linear. Log sets the current axis to logarithmic. You cannot put the axis into a log scale if either end of the axis range is zero or negative.
`Trace Variable`	Modify the trace variable. Click the Trace Variable button, then select a variable from the list box. Click OK to accept your changes, or click Cancel to disregard. The trace names show which trace variable is currently used.

Encrypting a Library

To encrypt a library:

1. Choose *File – Encrypt Library* to open the Library Encryption dialog box.
2. In the *Library to be encrypted* text box, enter the path to the library file to be encrypted.
3. In the *Encrypted Library Folder* text box, enter the path to the folder where the encrypted library should be kept.
4. Select *Partial Encryption* if you want to encrypt the library partially.

To partially encrypt a library, put the identifier `$CDNENCSTART` at the beginning and the identifier `$CDNENCFINISH` at the end of the text to be encrypted.

5. Select *Show Interfaces* to encrypt only model text and show the interfaces. The Show Interfaces check box is not available if Partial Encryption is selected.
6. Click *OK*.

   The encrypted library is placed in the folder specified in the Encrypted Library Folder text box.

Note: The encryption utility does not check for syntax error in a library.

Model Import Wizard

You use the Model Import wizard to:

- associate symbols for the simulator models that could not be recognized automatically
- update existing symbols for the simulator models

Choose *File – Model Import Wizard* to start the wizard.

The following sections explains the different pages of the wizard.

Specify Library Page

`Enter Input Model Library`	Specify the name and the location of the `.lib` file containing the models for which symbols are required.
`Enter Destination Symbol Library`	Specify the name and the location of the symbol library (`.olb`) to be created.

Associate /Replace Symbols Page

`Models with symbol`	Select this check box, to list the models, for which the Model Import Wizard can find matching symbols, along with the corresponding symbol names.
`Models without symbol`	Select this check box, to list the models for which matching symbols could not be found.

`Associate/Replace Symbols`	Select this toggle button when you want to attach an existing symbol to the selected simulation model. *Associate Symbol* button is available when you select a model from the *Models without symbols* list. Use the *Associate Symbol* button to associate an existing symbol to a model without symbol *Replace Symbol* button is available when the selected model already has a symbol associated to it. Use the *Replace Symbol* button to replace a symbol associated to a model, by an existing symbol of your choice.
`Back`	This button is available only when the Model Import wizard is launched using the *Model Import Wizard* command from the *File* drop-down menu. Use this button to move back to the *Specify Library* page of the wizard.
`Finish`	Select this if you want to stop the process of creating symbols. In case you have models that do not have any symbols associated to them, a message box appears, asking you whether rectangular symbols should be attached to these models or not.
`View Model Text`	Displays the model text for the selected model in a new window.

Select Matching Page

`Select a library containing matching symbols`	Specify the library that contains the symbol you want to associate the selected model. You can either use the browse button to navigate to the desired olb file, or select the file from the drop-down list box. The drop-down menu lists a maximum of 10 most recently used libraries.
`Matching Symbols`	Lists all the symbols that can be associated with the model selected in *Associate/Replace Symbol* page. The matching symbols list is generated based on model definition of the selected model by the user.
`Symbol pane`	Displays the graphical shape of the symbol currently selected in the *Matching Symbols* list.

`Next`	Select this to move to the next step.
`Cancel`	Select this to cancel the process of associating an existing symbol to a simulation model.
`View Model Text`	Displays the model text for the selected model in a new window.

Define Pin Mapping Page

Use this page for pin to port mapping between the selected symbol shape and the model definition.

While you complete the pin-port mapping, you can view the symbol shape in the *Symbol* pane on the right of the wizard, and the use the *View Model Text* button to view the model definition.

Note: All the symbol pins must be mapped to a model terminal. After you have mapped each symbol pin to a unique model terminal, if there are any optional model terminals left, you may leave them unmapped.

`Model Terminal`	Lists the port names from the model definition
`Symbol Pin`	Lists the symbol pin names. From the drop-down list, select the pin name that is to be associated with the listed model terminal.
`Optional Model Terminals`	Lists the optional ports in the model definition. Depending on the availability of symbol pins, you may or may not map these.
`Symbol window`	Displays the shape of the symbol selected in the *Matching Symbols* list.
`Back`	Select this to move to the previous step, where you selected a matching symbol.
`Cancel`	Select this to cancel the process of associating an existing symbol to a simulation model.
`Save Symbol`	Select this to complete the process of associating a symbol to the selected model and to jump back to the *Associate/ Replace Symbol* page.
`View Model Text`	Displays the model text for the selected model in a new window.

Model Editor Reference Information

The topics covered in this chapter are:

Diode spec entry

- Diode: Template-based Model Parameters
- Diode: forward current
- Diode: junction capacitance
- Diode: reverse leakage
- Diode: reverse breakdown
- Diode: reverse recovery
- Diode: Test Node Mapping

Diode: Template-based Model Parameters

Following table lists the template-based model parameters for diodes:

Model Parameters	Description	Unit	Default
AF	flicker noise exponent		1
BV	reverse breakdown knee voltage	V	100
CJO	Junction capacitance	F	0
EG	Activation energy	eV	1.11
FC	Depletion capacitance coefficient		0.5
IBV	reverse breakdown knee current	A	0.001
IS	saturation current	A	10f
KF	flicker noise coefficient		0
M	Grading coefficient		0.5
N	emission coefficient		1
RS	source ohmic resistance	ohm	0
TT	transit time	sec	0
VJ	Junction potential	V	1
XTI	IS temperature exponent		3

Diode: forward current

The model spec for diode forward current estimates the parameters IS and RS from three voltage and current values. Try to include data from low current values (where the increase in current is exponential), moderate current values, and high current value (where the increase in current is clearly resistive).

Device curve

`Vfwd`	forward voltage across junction for Ifwd
`Ifwd`	Forward current @ Vfwd

The model parameters XTI and EG can be changed. They are set to be typical values for silicon diodes. For Schottky-barrier diodes, these may be changed to XTI=2 and EG=0.69, which will give better modeling over temperature.

Also, it is sometimes helpful to set up traces for a few values of temperature (use the Trace command). For a better fit at different temperatures, adjust XTI.

Diode: junction capacitance

The model spec for diode junction capacitance estimates the parameters CJO and M from capacitance values given at non-zero reverse biases (a zero value for a Vj data point is OK).

Device curve

`Vrev`	reverse voltage across diode (junction) for Cj
`Cj`	junction capacitance @ Vrev

The value for FC are set to be normal for silicon diodes, but is relatively unimportant, as forward capacitance is dominated by diffusion capacitance (and modeled by transit time).

Data sheets

The data sheets for most switching and power diodes have little detail about reverse bias capacitance because it is not too important. Varicap diodes usually have better, more complete information. Be aware that the diode package adds some fixed amount of capacitance that is not included in the device model, but may be included with a small capacitor across the diode. Having determined the package capacitance, subtract that from the total capacitance to model the diode junction.

Model

parameter	Model description	Default value
CJO	zero-bias junction capacitance	1E-12
VJ	junction potential	.75
M	junction grading coefficient	.3333
FC	onset of forward-bias depletion capacitance coefficient	.5

Diode: reverse leakage

The model spec for diode reverse leakage derives the generation-recombination current values for the device which, with capacitance modeling (previous screen), provides the primary leakage mechanism of the diode junction.

Reverse current leakage is increased by imperfections in manufacturing, which are not modeled. Breakdown also increases reverse current. See Diode: reverse breakdown for more information.

Device curve

Vrev	reverse voltage for Irev	
Irev	reverse (leakage) current @ Vrev	
Model parameter	Model description	Default value
ISR	recombination current saturation value	1E-10
NR	recombination current emission coefficient	2

Note: By setting the ISR and NR parameters, the forward current of this model may need to be re-extracted to ensure accuracy.

Diode: reverse breakdown

The model spec for diode reverse breakdown estimates the parameters BV and IBV for reverse breakdown operation, which is how voltage regulator (Zener or avalanche) diodes work. Enter the values for Vz, Iz, and Zz.

Device data

`Vz`	nominal Zener voltage @ Iz
`Iz`	nominal Zener current for Vz
`Zz`	Zener impedance (resistance) @ Vz,Iz

BV and IBV will nearly equal Vz and Iz. As the breakdown effect is modeled by an exponential function, the value of BV and IBV will adjust so that device impedance, Zz (ratio of the change in voltage to the change in current) is correct at Vz,Iz.

Model

`parameter`	Model description	Default value
`BV`	reverse breakdown voltage (a positive value)	100
`IBV`	reverse breakdown current (a positive value)	1E-4

Diode: reverse recovery

The model spec for diode reverse recovery shows a transient simulation of the diode switching. Some of the parameters that have dynamic effects (e.g., CJO) are included in the simulation. Adjust the X axis if you need to see the entire waveform.

Device data

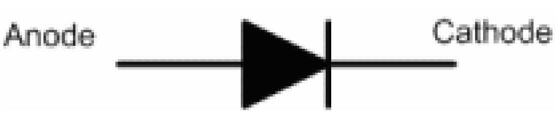
`Trr`	reverse recovery time
`Ifwd`	forward current (before switching)
`Irev`	initial reverse current
`Rl`	load resistance (total load of test fixture)

The model spec for diode reverse recovery also estimates the parameter TT from switching time. Enter values for the above list. Make sure to include the test fixture resistance and pulse generator resistance in Rl.

Model

Model parameter	Model description	Default value
TT	transit time	5E-9

Diode: Test Node Mapping



Node	Stands for…	In the diagram…
TERM_AN	Forward Current terminal	Is the current through Anode
NODE_AN	Anode Voltage Node	Is the voltage at Anode
NODE_CAT	Cathode Voltage Node	Is the voltage at Cathode

BJT spec entry

- Bipolar transistor: Template-based Model Parameters
- Bipolar transistor: junction voltage
- Bipolar transistor: output admittance
- Bipolar transistor: forward DC beta
- Bipolar transistor: VCE(sat) voltage
- Bipolar transistor: C-B capacitance
- Bipolar transistor: E-B capacitance
- Bipolar transistor: storage time
- Bipolar transistor: gain bandwidth

- Bipolar Junction Transistor: Test Node Mapping

Bipolar transistor: Template-based Model Parameters

Following table lists the template-based model parameters for bipolar transistor:

Model parameters	Description	Units	Default
AF	flicker noise exponent		1.0
BF	ideal maximum forward beta		100.0
BR	ideal maximum reverse beta		1.0
CJC	base-collector zero-bias p-n capacitance	F	0.0
CJE	base-emitter zero-bias p-n capacitance	F	0.0
CJS (CCS)	substrate zero-bias p-n capacitance	F	0.0
EG	bandgap voltage (barrier height)	eV	1.11
FC	forward-bias depletion capacitor coefficient		0.5
IKF (IK)	corner for forward-beta high-current roll-off	A	10
IKR	corner for reverse-beta high-current roll-off	A	100MEG
IRB	current at which Rb falls halfway to	A	100MEG
IS	transport saturation current	A	1f
ISC (C4) †	base-collector leakage saturation current	A	1E-15
ISE (C2) †	base-emitter leakage saturation current	A	1E-13
ITF	transit time dependency on Ic	A	0.0
KF	flicker noise coefficient		0.0
MJC (MC)	base-collector p-n grading factor		0.33
MJE (ME)	base-emitter p-n grading factor		0.33
MJS (MS)	substrate p-n grading factor		0.0
NC	base-collector leakage emission coefficient		2.0
NE	base-emitter leakage emission coefficient		1.5
NF	forward current emission coefficient		1.0

Model parameters	Description	Units	Default
NR	reverse current emission coefficient		1.0
PTF	excess phase @ 1/(2π·TF)Hz	degree	0.0
RB	zero-bias (maximum) base resistance	ohm	0.0
RBM	minimum base resistance	ohm	RB
RC	collector ohmic resistance	ohm	0.0
RE	emitter ohmic resistance	ohm	0.0
TF	ideal forward transit time	sec	0.0
TR	ideal reverse transit time	sec	0.0
VAF (VA)	forward Early voltage	V	100MEG
VAR (VB)	reverse Early voltage	V	100MEG
VJC (PC)	base-collector built-in potential	V	0.75
VJE (PE)	base-emitter built-in potential	V	0.75
VJS (PS)	substrate p-n built-in potential	V	0.75
VTF	transit time dependency on Vbc	V	100MEG
XCJC	fraction of CJC connected internally to Rb		1.0
XTB	forward and reverse beta temperature coefficient		0.0
XTF	transit time bias dependence coefficient		0.0
XTI (PT)	IS temperature effect exponent		3.0

Bipolar transistor: junction voltage

The model spec for the bipolar transistor Vbe(sat) voltage estimates the parameter IS, RB, and NF from the saturation characteristics of the transistor. IS is a semiconductor junction parameter and should not be confused with the collector current in saturation. The data sheet will have values or curves for Vbe in a "forced beta" (where the ratio Ic/Ib is much lower than the normal current gain) or "saturated" condition. Enter the values of Vbe versus Ic. Also, be sure to check/enter the value for the “forced beta” ratio of collector to base current.

The last two model parameters, XTI and EG, may be changed. We have set them to be normal values for silicon transistors.

Device data

`Ic`	collector current for Vbe
`Vbe`	base-emitter voltage @ Ic (device in saturation)

Conditions

`Ic/Ib`	"forced beta" ratio for device curve

Model

`parameter`	Model description	Default value
`IS`	saturation current	1E-14
`RB`	base resistance	1E-3
`NF`	forward current emission coefficient	1
`XTI`	temperature coefficient for IS	3
`EG`	activation energy	1.11

Bipolar transistor: output admittance

The model spec for the bipolar transistor output admittance estimates the parameter VAF, which sets the output conductance of the transistor in a common emitter configuration.

The parameter VAF controls one aspect of basewidth modulation in the Gummel-Poon transistor model. This manifests itself as output conductance. Typical values are 50 to 100 volts for normal transistors and 1 to 10 volts for super-beta transistors.

Device curve

`Ic`	collector current for hoe
`hoe`	small-signal open-circuit output admittance @Ic (and @Vce)

Conditions

`Vce`	collector-emitter voltage for the device curve

Model

`parameter`	Model description	Default value
`VAF`	forward early voltage	100

Bipolar transistor: forward DC beta

The model spec for Bipolar Transistor Forward DC Beta estimates parameters for the celebrated Gummel-Poon bipolar transistor model. Try to include data from low current values (beta rising), moderate current values, and high current value (beta falling). The value Vce adjusts the beta data for basewidth modulation effects.

Data sheets

Transistor data sheets usually show minimum beta values and have a maximum value for only one collector current value. One way to obtain an average value is to use the current level that specifies both minimum and maximum beta, using a value somewhat below the average of the minimum and maximum. Then ratio the other minimum values by the same amount. Or just use the curves (if available) from the data sheet.

The value for XTB has been set to be zero for bipolar transistors but may be changed. It is sometimes helpful to set up traces for a few values of temperature (use Trace command) for adjusting XTB.

Device curve

Ic	collector current for hFE (@ Vce)
hFE	forward DC beta @ Ic

Conditions

Vce	collector-emitter voltage for the device curve

Model

parameter	Model description	Default value
BF	ideal maximum forward beta	100
ISE	non-ideal base-emitter diode saturation current	0
NE	non-ideal base-emitter diode emission coefficient	1.5
IKF	forward beta roll-off “knee” current	0
NK	forward beta roll-off slope exponent	.5
XTB	forward beta temperature coefficient	1.5

Bipolar transistor: VCE(sat) voltage

The model spec for the bipolar transistor Vce(sat) voltage estimates more parameters for the celebrated Gummel-Poon transistor model. Try to include data from low current values (Vce falling), moderate current values, and high current value (Vce rising). Also, be sure to check/ enter the value for the "forced beta" ratio of collector to base current.

The reverse Gummel-Poon parameters correspond to the forward parameters, except they are for reverse operation (that is, emitter swapped with the collector). It would be more accurate to obtain these the same way as the forward parameters, but reverse operation is rarely published data. Fortunately, it does not affect operation when the transistor is saturated, that is, when the base-collector junction is forward biased.

Device curve

`Ic`	collector current for Vce (@ Ib/Ic)
`Vce`	collector-emitter voltage @ Ic

Conditions

`Ic/Ib`	"forced beta" ratio for device curve

Model

`parameter`	Model description	Default value
`BR`	ideal maximum reverse beta	1
`ISC`	non-ideal base-collector diode saturation current	0
`NC`	non-ideal base-collector diode emission coefficient	2
`IKR`	reverse beta roll-off “knee” current	0
`RC`	series collector resistance	0

Bipolar transistor: C-B capacitance

The model spec for the bipolar transistor C-B capacitance estimates the parameters CJC and MJC from capacitance values given at non-zero reverse biases (a zero value for Vcb is OK).

The value for FC has been set to be normal for silicon transistors but may be changed. The value of FC is relatively unimportant, as forward capacitance is dominated by diffusion capacitance (and modeled by transit time).

Be aware that the transistor package adds some fixed amount of capacitance that is not included in the device model, but may be included by the user with a small capacitor across the junction. Having determined the package capacitance, subtract that from the total capacitance to model the junction.

Device curve

`Vcb`	reverse voltage collector-base junction for Cobo
`Cobo`	open circuit output capacitance @ Vcb

Model

`parameter`	Model description	Default value
`CJC`	zero-bias collector-base junction capacitance	2E-12
`VJC`	collector-base junction potential	.75
`MJC`	collector-base junction grading coefficient	.33
`FC`	coefficient for onset of forward-bias depletion capacitance	.5

Bipolar transistor: E-B capacitance

The model spec for the bipolar transistor E-B capacitance estimates the parameters CJE and MJE from capacitance values given at non-zero reverse biases (a zero value for Veb1 is OK).

The value of FC from the bipolar transistor C-B capacitance is used and is still relatively unimportant, as forward capacitance is dominated by diffusion capacitance.

Be aware that the transistor package adds some fixed amount of capacitance that is not included in the device model, but may be included by the user with a small capacitor across the junction. Having determined the package capacitance, subtract that from the total capacitance to model the junction.

Device curve

`Veb`	reverse voltage emitter-base junction for Cibo
`Cibo`	open circuit input capacitance @ Veb

Model

`parameter`	Model description	Default value
`CJE`	zero-bias emitter-base junction capacitance	2E-12
`VJE`	emitter-base junction potential	.75
`MJE`	emitter-base junction grading coefficient	.33

Bipolar transistor: storage time

The model spec for the bipolar transistor storage time estimates the parameter TR, which controls the delay until the transistor leaves saturation when switching off. Be sure to check/ enter a value for the "forced beta" ratio when the transistor was on and saturated.

The storage time curve is controlled by the forward and reverse beta characteristics of the transistor. The parameter TR acts like a multiplying factor without changing the character of the curve. Use the storage time for the collector current range you are interested in.

Device curve

`Ic`	collector current for ts (@ Ic/Ib)
`ts`	storage time (not "shelf life") @ Ic

Conditions

`Ic/Ib`	"forced beta" ratio for device curve

Model

`parameter`	Model description	Default value
`TR`	reverse transit time	10E-9

Bipolar transistor: gain bandwidth

The model spec for the bipolar transistor gain bandwidth estimates the parameter TF, which, along with collector-base capacitance, limits high-frequency gain. The value of TF also controls rise and fall times in switching circuits, which is another way to measure transistor speed, although there isn't a rule-of-thumb conversion between rise/fall time and high-frequency cutoff.

Tip

It is sometimes helpful to set up traces for a few values of Vce for adjusting VTF.

Device curve

`Ic`	collector current for fT (@ Vce)
`fT`	frequency at which small-signal forward current transfer ratio extrapolates to unity @ Ic

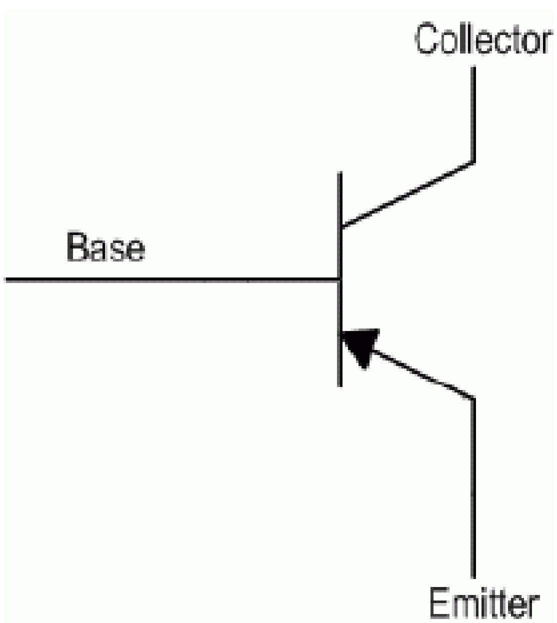
Conditions

`Vce`	collector-emitter voltage for device curve

Model

`parameter`	Model description	Default value
`TF`	forward transit time	10E-9
`ITF`	current for TF dependency on Ic	1
`XTF`	coefficient for TF dependency on Vce	10
`VTF`	voltage for TF dependency on Vce	10

Bipolar Junction Transistor: Test Node Mapping



Node	Stands for…	In the diagram…
TERM_IC	Collector Current terminal	Is the current through Collector
TERM_IB	Base Current terminal	Is the current through Base
NODE_VC	Collector Voltage node	Is the voltage at Collector
NODE_VB	Base Voltage node	Is the voltage at Base
NODE_VE	Emitter Voltage node	Is the voltage at Emitter

Darlington transistor spec entry

Darlington transistor: ON voltage

The model spec for the Bipolar Darlington Transistor Vbe(ON) vs. Ic is used to estimate the parameters IS and RB from the ON or linear (not saturation) region of the transistor. IS is a semiconductor junction parameter and should not be confused with the collector current in saturation.

The data sheet has values or curves for Vbe(on) at a given Vce condition. Enter the values of Vbe(ON) vs. Ic. Also, be sure to check the value for the collector-emitter ON voltage condition (Vce) and enter it if necessary.

Enter the values for R1 and R2. They represent the resistance across each of the base-emitter junctions. If they are not given, you can assume they are very large.

Device curve

`Ic`	collector current for Vbe
`Vbe`	base-emitter voltage @ Ic (device not in saturation)
`R1`	resistance across the base-emitter of the input transistor
`R2`	resistance across the base-emitter of the output transistor

Conditions

`Vce`	collector-emitter voltage for the device curve

Model

`parameter`	Model description	Default value
`IS`	saturation current	1E-12
`RB`	base resistance	1
`Area`	Darlington area factor	1
`R1`	resistance across the base-emitter of the input transistor	10000
`R2`	resistance across the base-emitter of the output transistor	1000

Darlington transistor: current gain

The model spec for the bipolar Darlington transistor forward current gain estimates parameters for the celebrated Gummel-Poon bipolar transistor model (BF, ISE, NE, IKF, NK).

Device curve

Try to include data from low current values (beta rising), moderate current values, and high current value (beta falling).

`Ic`	collector current for hFE (@ Vce)

`hFE`	forward DC beta @ Ic

Conditions

The value Vce adjusts the beta data for basewidth modulation effects.

`Vce`	collector-emitter voltage for the device curve

Model

`parameter`	Model description	Default value
`BF`	ideal maximum forward beta	1000
`ISE`	non-ideal base-emitter diode saturation current	1E-9
`NE`	non-ideal base-emitter diode emission Coefficient	2
`IKF`	forward beta roll-off “knee” current	0.1
`NK`	forward beta roll-off slope exponent	0.5

Darlington transistor: saturation voltage

The model spec for the bipolar Darlington transistor Vce(sat) voltage estimates parameters for the celebrated Gummel-Poon transistor model.

The reverse Gummel-Poon parameters correspond to the forward parameters, except that they are for reverse operation (that is, for the emitter instead of the collector). It would be more accurate to obtain these the same way as the forward parameters, but reverse operation data is rarely published.

Device curve

Try to include data from low current values (Vce falling), moderate current values, and high current values (Vce rising). Also, be sure to check/enter the value for the forced beta ratio of collector to base current.

`Ic`	collector current for Vce @ Ib/Ic

`Vce`	collector-emitter voltage @ Ic

Conditions

`Ic/Ib`	forced beta ratio for device curve

Model

`parameter`	Model description	Default value
`BR`	ideal maximum reverse beta	1
`ISC`	non-ideal base-collector diode saturation current	1E-13
`NC`	non-ideal base-collector diode emission coefficient	2
`IKR`	reverse beta roll-off knee current	0.1
`RC`	series collector resistance	0.1

Darlington transistor: input capacitance

The model spec for the bipolar Darlington transistor E-B capacitance estimates the parameters CJE, VJE and MJE from capacitance values given at non-zero reverse biases (a zero value for Veb1 is OK).

The transistor package adds some fixed amount of capacitance that is not included in the device model, but you can include it in the model by using a small capacitor across the junction. After determining the package capacitance, subtract that value from the total capacitance to model the junction.

Device curve

`Veb`	reverse voltage emitter-base junction for Cibo
`Cibo`	open circuit input capacitance @ Veb

Model

`parameter`	Model description	Default value
`CJE`	zero-bias emitter-base junction capacitance	1E-11
`VJE`	emitter-base junction potential	0.75
`MJE`	emitter-base junction grading coefficient	0.33

Darlington transistor: output capacitance

The model spec for the bipolar Darlington C-B capacitance estimates the parameters CJC, VJC and MJC from capacitance values given at non-zero reverse biases.

The transistor package adds some fixed amount of capacitance that is not included in the device model, but you can include it in the model by using a small capacitor across the junction. After determining the package capacitance, subtract that value from the total capacitance to model the junction.

Device curve

`Vcb`	reverse voltage collector-base junction for Cobo
`Cobo`	open circuit output capacitance @ Vcb

Model

`parameter`	Model description	Default value
`CJC`	zero-bias collector-base junction capacitance	1E-11
`VJC`	collector-base junction potential	0.75
`MJC`	collector-base junction grading coefficient	0.33

Darlington transistor: storage time

The model spec for the bipolar Darlington transistor storage time estimates the parameter TR, which controls the delay until the transistor leaves saturation when switching off.

The storage time curve is controlled by the forward and reverse beta characteristics of the transistor. The parameter TR acts like a multiplying factor without changing the character of the curve. Use the storage time for the collector current range you are interested in.

Device curve

Enter a single value for the storage time.

`ts`	storage time (not shelf life) @ Ic, Ib1 and Ib2

Conditions

Be sure to check values for the current conditions (Ic, Ib1 and Ib2) under which the specified storage time occurs and enter them if necessary.

`Ic`	collector current for ts (@ Ic, Ib1 and Ib2)
`Ib1`	base current to generate Ic

`Ib2`	base current to turn off Ic

Model

parameter	Model description	Default value
`TR`	reverse transit time	1E-7

Darlington transistor: rise time

The model spec for the bipolar Darlington transistor rise time estimates the parameter TF, which, along with the collector-base capacitance, controls switching time and limits high-frequency gain. Enter a single value for the rise time.

Device curve

`tr`	rise time @ Rl, Vcc and Ib

Conditions

Be sure to check values for the operating conditions (Vcc, Ib and Rl) under which the specified rise time occurs and enter them if necessary.

`Vcc`	supply voltage for collector load
`Ib`	step input current value
`Rl`	collector load resistance

Model

parameter	Model description	Default value
`TF`	forward transit time	1E-9

Darlington transistor: diode forward current

The model spec for diode forward current estimates the parameters IS and RS from three voltage and current values. Try to include data from low current values (where the increase in current is exponential), moderate current values, and high current value (where the increase in current is clearly resistive).

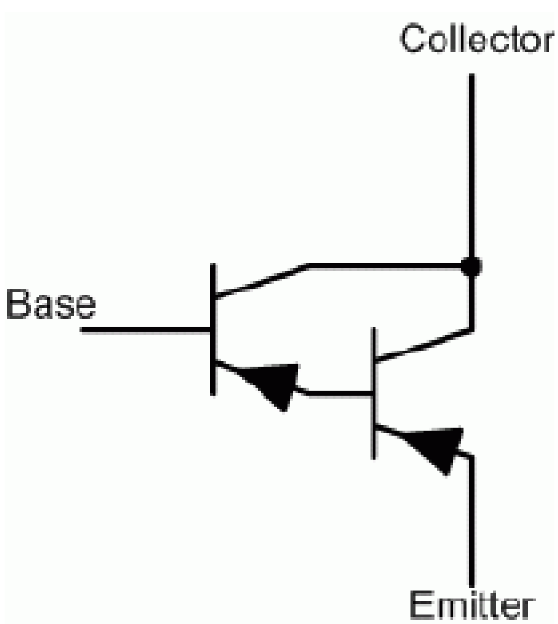
Device curve

`Vfwd`	forward voltage across junction for Ifwd
`Ifwd`	Forward current @ Vfwd

Model

`parameter`	Model description	Default value
`IS`	saturation current	1E-14
`N`	emission coefficient	1
`RS`	series resistance	1e-3
`XTI`	IS temperature exponent	3

Darlington Transistor: Test Node Mapping



`Node`	Stands for…	In the diagram…
`TERM_IC`	Collector Current terminal	Current through Collector
`TERM_IB`	Base Current terminal	Current through Base
`NODE_VC`	Collector Voltage node	Voltage at Collector
`NODE_VB`	Base Voltage node	Voltage at Base
`NODE_VE`	Emitter Voltage node	Voltage at Emitter

JFET spec entry

- Diode: Template-based Model Parameters
- JFET: transconductance
- JFET: output conductance
- JFET: transfer curve
- JFET: reverse transfer capacitance
- JFET: input capacitance
- JFET: Passive Gate leakage
- JFET: active gate leakage
- JFET: noise voltage
- Junction Field Emitter Transistor: Test Node Mapping

JFET: Template-based Model Parameters

Following table lists the template-based model parameters for JFET:

Model parameters	Description	Units	Default
AF	flicker noise exponent		1
BETA	transconductance coefficient	A/V^2	1.5m
CGD	zero-bias gate-drain *p-n* capacitance	F	0
CGS	zero-bias gate-source *p-n* capacitance	F	0
FC	forward-bias depletion capacitance coefficient		0.5
IS	gate *p-n* saturation current	A	10f
KF	flicker noise coefficient		0
LAMBDA	channel-length modulation	1/V	0
MODE	Device mode		1
PB	gate *p-n* potential	V	1.0
RD	drain ohmic resistance	ohm	0
RS	source ohmic resistance	ohm	0
VTO	threshold voltage	V	-2.0

JFET: transconductance

The model spec for JFET transconductance estimates the parameter BETA, which sets the change in drain current vs. gate-source voltage. BETATCE is set manually, using traces at other temperatures to judge the effect (the default setting is a nominal value chosen from inspecting many data sheets).

Device curve

`Id`	drain current for gFS
`gFS`	forward transconductance @ Id

Also, it is sometimes helpful to set up traces for a few values of temperature (use the Add Trace command from the Plot menu) for adjusting BETATCE.

Model

parameter	Model description	Default value
BETA	transconductance coefficient	1E-4
RD	drain resistance	1
RS	source resistance	1
BETATCE	temperature coefficient for BETA	-0.5

JFET: output conductance

The model spec for JFET output conductance estimates the parameter LAMBDA, which sets the slope of the drain-current vs. drain-source voltage in saturation.

Device curve

Id	drain current for gOS
gOS	output conductance @ Id

Model

parameter	Model description	Default value
LAMBDA	channel-length modulation	1E-6

JFET: transfer curve

The model spec for JFET transfer curve estimates the parameter VTO, which is the threshold (or pinchoff) voltage.

Device curve data

`Vgs`	gate-source voltage for Id (@ Vds)
`Id`	drain current @ Vgs

Conditions

`Vds`	drain-source voltage for device curve

You can control the temperature dependence of this model curve with the parameter VTOTC.

Model

`parameter`	Model
`description`	Default

value

`VTO`	threshold voltage	-2
`VTOTC`	temperature coefficient for VTO	-2.5E-3

Note: The SPICE standard is for VTO to be a negative value for a depletion transistor, regardless of device type (NJF or PJF).

JFET: reverse transfer capacitance

The model spec for JFET reverse transfer capacitance estimates the parameters CGD and M. The reverse transfer, or "Miller," capacitance is modeled.

Device curve

`Vgs`	gate-source voltage for Crss (@ Vds)
`Crss`	reverse transfer capacitance @ Vgs

Conditions

`Vds`	drain-source voltage for device curve

The parameter FC applies to forward-biased junctions and is included for completeness.

Model

`parameter`	Model description	Default value
`CGD`	zero-bias gate-drain capacitance	1E-12
`M`	junction grading factor	.5
`PB`	built-in potential	1
`FC`	forward-bias coefficient	.5

JFET: input capacitance

The model spec for JFET input capacitance estimates the parameter CGS, which is derived from the difference between Ciss and Crss. As a check, since most JFETs are designed to be symmetrical, the value found for CGS should be close to that found for CGD. See JFET: reverse transfer capacitance for more information.

Device curve data

`Vgs`	gate-source voltage for Ciss (@ Vds)
`Ciss`	input capacitance @ Vgs

Conditions

`Vds`	drain-source voltage for device curve

Model

`parameter`	Model description	Default value
`CGS`	zero-bias gate-source capacitance	1E-12

JFET: Passive Gate leakage

The model spec for JFET passive gate leakage derives the generation-recombination current values for the device, which, with the capacitance modeling (previous screens), provides the primary leakage mechanism of the device's junction.

Passive reverse current leakage is increased by imperfections in manufacturing and breakdown, which are not modeled.

Device Curve

`Vdg`	drain-gate voltage for Igss
`Igss`	gate leakage current @ Vdg

It is sometimes helpful to set up traces for a few values of temperature (use the Add Trace command from the Plot menu) for adjusting XTI.

Model

parameter	Model description	Default value
ISR	recombination current saturation value	0
NR	recombination current emission coefficient	2
IS	junction saturation current	1E-14
N	junction emission coefficient	1
XTI	IS temperature coefficient	3

JFET: active gate leakage

The model spec for JFET active gate leakage estimates active gate current when the JFET is on, which may be much larger than when the JFET is cut off.

Impact ionization by drain-current carriers generate carriers in the gate space-charge region, which get swept out through the gate. This causes gate current which is an exponential function of drain voltage and proportional to drain current.

Note that the lowest values of active leakage current are generally less than the passive leakage values; this is because the passive values are measured with source and drain shorted together, which usually doubles the junction area and, thus, the current. Active leakage current occurs in the drain-gate junction only, so the lowest levels represent passive leakage for that junction.

Device curve

`Vdg`	drain-gate voltage for Ig (@ Id)
`Ig`	gate leakage current @ Vdg

Conditions

`Id`	drain current for device curve

Model

`parameter`	Model description	Default value
`ALPHA`	impact ionization coefficient	1E-6
`VK`	ionization “knee” voltage	1

JFET: noise voltage

The model spec for JFET noise voltage estimates the parameter KF, to set the correct amount of flicker noise. AF may be set manually but is normally close to 1. The broadband noise of a JFET is shot noise and is set by the conductance of the channel.

Device curve

`Freq`	frequency for en (@ Ids)
`en`	equivalent input noise voltage (in volts/root-hertz) @ Freq

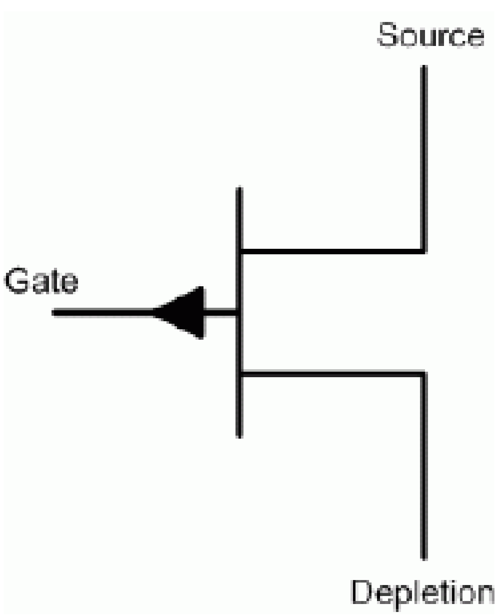
Conditions

`Ids`	drain current for device curve

Model

`parameter`	Model description	Default value
`KF`	flicker noise coefficient	1E-18
`AF`	flicker noise exponent	1

Junction Field Emitter Transistor: Test Node Mapping



`Node`	Stands for…	In the diagram…

`TERM_ID`	Depletion Current terminal	Current through Drain
`TERM_IG`	Gate Current terminal	Current through Gate
`NODE_VD`	Depletion Voltage node	Voltage at Drain
`NODE_VG`	Gate Voltage node	Voltage at Gate
`NODE_VS`	Source Voltage node	Voltage at Source

Power MOSFET spec entry

- Power MOSFET: Template-based Model Parameters
- MOSFET transconductance
- MOSFET transfer curve
- MOSFET Rds (on) resistance
- Power MOSFET: zero-bias leakage
- Power MOSFET: turn-on charge
- Power MOSFET: output capacitance
- Power MOSFET: switching time
- Power MOSFET: reverse drain current
- MOSFET: Test Node Mapping

Power MOSFET: Template-based Model Parameters

Following table lists the template-based model parameters for Power MOSFET:

Parameter	Description	Unit	Default
BVD	Reverse Breakdown Voltage	V	10MEG
CGDC	Cross compensating capacitance	F	1E-12
CGDO	gate-drain overlap capacitance/channel width	F/m	1E-18

Parameter	Description	Unit	Default
CGSO	gate-source overlap capacitance/channel width	F/m	1E-18
CJOC	Cross junction capacitance	F	0
EGD	Activation energy	eV	1.11
FCD	Depletion capacitance coefficient		0.5
GAMMA	Body-effect parameter	$V^{1/2}$	0
IBVD	current at voltage breakdown	A	1E-10
ISD	Saturation current	A	1E-14
KP	transconductance coefficient	A/V^2	20u
LAMBDA	Channel-length	1/V	1E-10
LD	Drain inductance	H	5E-9
LS	Source inductance	H	12.5E-9
MC	Cross grading coefficient		0
MD	Cbd grading coefficient		0.5
MD2	Cbd grading coefficient for Mtype=2		0.5
ND	Emission coefficient		*1*
RD	Drain ohmic resistance	Ohm	0.2
RDS	Body resistance	Ohm	2E6
RG	Gate resistance	Ohm	12
RG2	Gate resistance for Mtype=2	Ohm	12
RLD	LD damping resistance	Ohm	100
RLS	LS damping resistance	Ohm	100
RS	source ohmic resistance	Ohm	0.01
RSD	Ohmic resistance	Ohm	0
TTD	Transit time	sec	0
VJC	Cross junction potential	V	0.75
VJD	Cbd junction potential	V	1
VJD2	Cbd junction potential for Mtype=2	V	1

Parameter	Description	Unit	Default
VTO	Threshold voltage	V	0
XJC	Fraction (0<=XJC<1)		0
XTID	Junction current temperature exponents for drain junctions		3

MOSFET transconductance

The model spec for power MOSFET transconductance estimates the basic geometry of the power MOSFET, its conductance parameter, and high-current effects of series resistance in the device.

Device curve

`Id`	drain current for gFS
`gFS`	forward transconductance @ Id

Many general assumptions are made about the device structure (such as oxide thickness), but the model will remain accurate in spite of these assumptions. The transconductance would ideally increase proportional to the square-root of the drain current, but is limited by the effects of RS.

Model

`parameter`	Model description	Default value
`KP`	transconductance	2E-5
`W`	channel width	.5
`L`	channel length	2E-6
`RS`	source ohmic resistance	10E-3

MOSFET transfer curve

The model spec for power MOSFET transfer curve estimates the device threshold voltage.

The actual value of VTO is not as important as obtaining a good value of drain current vs. Vgs as the device will be used. For library use, use a drain current close to the maximum continuous rating.

Device curve

`Vgs`	gate-source voltage for Id
`Id`	drain current @ Vgs

Model

`parameter`	Model
`description`	Default

value

`VTO`	zero-bias threshold voltage	3

MOSFET Rds (on) resistance

The model spec for power MOSFET Rds resistance estimates the "on-resistance" of the device.

The MOS model has three contributions to the "on-resistance": the channel resistance of the device, and an ohmic resistance in series with each the source and the drain. This model spec adjusts RD so the total resistance is correct. However, RD cannot become negative. Rds should be taken at an Id value not to exceed the absolute maximum rating for continuous current.

Device data

`Id`	drain current for Rds
`Vgs`	gate-source voltage for Rds
`Rds`	static drain-source on-state resistance @ Id and Vgs

Model

`parameter`	Model description	Default value
`RD`	ohmic drain resistance	10E-3

Power MOSFET: zero-bias leakage

The model spec for power MOSFET zero-bias leakage estimates the drain-source leakage of the device. This leakage is due primarily to surface effects and is modeled by a shunt drain-source resistance. Enter the values for the upper list.

Device data

`Vds`	drain-source voltage for Idss
`Idss`	zero gate voltage drain current @ Vds

Model

`parameter`	Model description	Default value
`RDS`	drain-source shunt resistance (simulator extension MOS model)	1E6

Power MOSFET: turn-on charge

The model spec for power MOSFET turn-on charge estimates the device's stray capacitances associated with the gate. These capacitances, along with the channel capacitance, make up the amounts of charge required to switch the device.

The value Qgs is the amount of charge required to raise the gate-source voltage from zero to that required to support the load current. Qgd is due to "Miller", or gate-drain, capacitance.

Device data

`Qgd`	gate-drain charge to switch load, Id, using supply, Vdd
`Qgs`	gate-source charge to start switching
`Vds`	supply voltage for Qgd (drain source)
`Id`	load (drain) current

Note that the values of CGSO and CGDO are multiplied by the channel width to yield the actual value of the capacitance.

Model

`parameter`	Model description	Default value
`CGSO`	gate-source overlap capacitance	4E-11
`CGDO`	gate-drain overlap capacitance	1E-11

Power MOSFET: output capacitance

The model spec for power MOSFET output capacitance estimates the output capacitance of the device.

The output capacitance is usually not critical, being small enough when compared with the load currents that are controlled by the device.

Device data

`Coss`	output capacitance @ Vds
`Vds`	drain-source voltage for Coss

Model

`parameter`	Model description	Default value
`CBD`	zero-bias bulk-drain junction capacitance	1E-9
`PB`	bulk junction potential	.8
`MJ`	bulk junction grading coefficient	.5
`FC`	bulk junction forward-bias capacitance coefficient	.5

Power MOSFET: switching time

The model spec for power MOSFET switching time estimates the value of series gate resistance from switching time.

Most power MOSFET devices use a self-aligned process with polysilicon gate material. The polysilicon impedes the gate current, reducing the charging rate of the gate, which increases the turn-on time. While there are many switching times specified (turn-on delay, rise time, etc.), they are all related by the parasitic capacitances, which have already been determined in the "gate charge" screen. Only the series resistance needs to be determined, which can be done reliably with the fall time characteristic.

Note that "fall time" means the period in which the drain current is "falling" in value, not the output voltage.

Device data

`tf`	fall time for switching load, Id, using supply, Vdd
`Id`	load (drain) current for tf
`Vdd`	supply voltage for tf
`Zo`	input generator impedance

Model

`parameter`	Model description	Default value
`RG`	gate ohmic resistance	5

Power MOSFET: reverse drain current

The model spec for power MOSFET reverse drain current estimates the forward voltage drop of the "body" diode.

The actual value of IS is not so important as obtaining a good value of voltage drop vs. current as the device will be used.

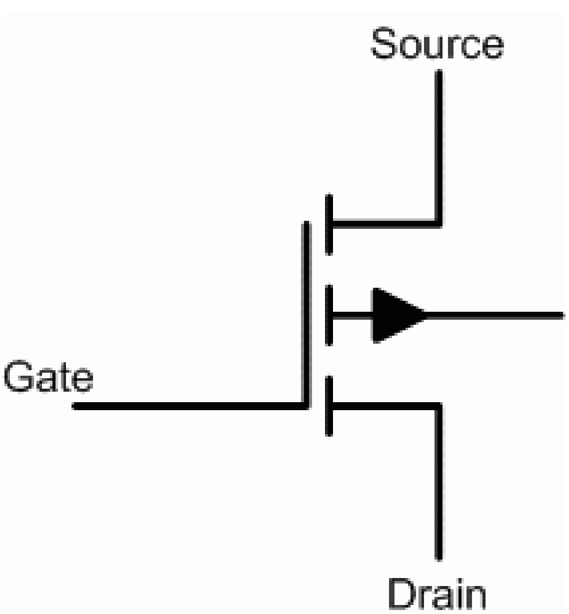
Device curve

`Vsd`	diode (source-drain) forward voltage for Idr
`Idr`	reverse drain current @ Vsd

Model

`parameter`	Model description	Default value
`IS`	bulk junction saturation current	1E-14
`N`	bulk junction emission coefficient	1
`RB`	bulk series resistance	1E-3

MOSFET: Test Node Mapping



`Node`	Stands for…	In the diagram…
`TERM_ID`	Drain Current terminal	Current through Drain
`TERM_IG`	Gate Current terminal	Current through Gate

`NODE_VD`	Drain Voltage node	Voltage at Drain
`NODE_VG`	Gate Voltage node	Voltage at Gate
`NODE_VS`	Source Voltage node	Voltage at Source

IGBT spec entry

There are limitations to IGBT spec entry:

- ❑ The Model Editor does not model ultra-fast IGBTs with buffer layer.
- ❑ The Model Editor does not include reverse free-wheel diodes (FREDs).

See the following topics for more detailed information:

- IGBT: Template-based Model Parameters
- IGBT: fall time
- IGBT: gate charge
- IGBT: saturation characteristics
- IGBT: transfer characteristics
- Insulated Gate Bipolar Transistor: Test Node Mapping

References

[1] A.R. Hefner, Jr., “INSTANT - IGBT Network Simulation and Transient Analysis Tool,” National Institute of Standards and Technology Special Publication 400-88, June 1992.

[2] A.R. Hefner, Jr., “An Investigation of the Drive Circuit Requirements for the Power Insulated Gate Bipolar Transistor (IGBT),” IEEE Transactions on Power Electronics,” Vol. 6, No. 2, April 1991, pp. 208-219.

[3] A.R. Hefner, Jr., “Modeling Buffer Layer IGBTs for Circuit Simulation,” IEEE Transactions on Power Electronics, “ Vol. 10, No. 2, March 1995, pp. 111-123.

IGBT: Template-based Model Parameters

Following table lists the template-based model parameters for IGBT:

Model parameters	Description	Units	Default
BF	maximum forward beta		9.0
CGSO	G-S overlap capacitance	F/m	0.0
CJC	B-C depletion capacitance	F	1.0u
CJE	B-E depletion capacitance	F	1.0u
CJGD	Cross junction capacitance	F	1.00E-09
EG	IS temperature energy gap	eV	1.1
IS	Saturation current	A	0.1f
KP	MOS transconductance	A/V2	20u
LAMBDA	channel-length modulation	1/V	1E-10
LE	emitter lead inductance	H	12.5E-9
MJC	B-C grading coefficient		0.33
MJE	B-E grading coefficient		0.33
MJGD	cross grading coefficient		0.0
NF	current forward emission coefficient		1
RB	base resistance	Ohm	0
RD	drain ohmic resistance	Ohm	0
RER	emitter lead resistance	Ohm	1E-14
RG	gate resistance	Ohm	0.1
RLE	LE damping resistance	Ohm	100.0
RS	source ohmic resistance	Ohm	0
TF	forward transit time	sec	0
VJC	B-C built-in potential	V	0.75

Model parameters	Description	Units	Default
`VJE`	B-E built-in potential	`V`	`0.75`
`VJGD`	cross junction potential	`V`	`0.5`
`VTO`	Threshold voltage	`V`	`0`
`XJGD`	Fraction of VJGD (0<=XJGD<=1)		`0`
`XTB`	Beta temperature exponent		`0`
`XTI`	IS temperature exponent		`3`

IGBT: fall time

The model spec for IGBT fall time shows the fall time of the collector current measured with inductive load at turn off. The initial collector current is modeled by Ic. At turnoff, this current falls rapidly, followed by a slow decaying tail. The rate of decay is controlled by the recombination rate of excess carriers in the lightly-doped epitaxial base layer. This recombination rate, in turn, is described by the base lifetime parameter TAU.

The maximum collector current, Icmax, and the maximum collector-emitter breakdown voltage, BVces, are obtained in data sheets in the absolute maximum ratings table.

Device data

`Icmax`	absolute maximum continuous collector current, in amps, at 25∘C
`Bvces`	absolute maximum collector-emitter breakdown voltage, in volts, with gate-emitter shorted
`tf`	collector current fall time, in seconds, with inductive load at the given Ic and Vce
`Ic`	collector current, in amps, at which tf is measured
`Vce`	collector-emitter voltage, in volts, at which tf is measured

Model

`parameter`	Model description	Default value
`AREA`	device active area, in square meters	1E-5
`TAU`	base lifetime, in seconds	7.1E-6
`WB`	metallurgical base width, in meters	90E-6

IGBT: transfer characteristics

The model spec for IGBT transfer characteristics displays the transfer characteristics at nominal temperature as the gate-emitter voltage increases from zero volts.

Data sheets

Data sheets usually provide the transfer characteristics curve. Points (Vge, Ic) should be sampled along the entire region of the curve. Care should be taken when sampling points near the threshold region as they will affect the accuracy of the parameter VT.

Device data

`Vge`	gate-emitter voltage, in volts, at 25◦?C at which the transfer characteristics are measured
`Ic`	collector current at the given Vge, in amps, at 25◦?C at which the transfer characteristics are measured
`Vce`	collector-emitter voltage, in volts, at which Vge and Ic are measured

Model

`parameter`	Model description	Default value
`KP`	MOSFET transconductance, in amps/(square volt)	.38
`VT`	internal MOSFET channel threshold voltage, in volts	2

IGBT: saturation characteristics

The model spec for IGBT saturation characteristics shows the saturation characteristics at nominal temperature as the collector current increases from zero amps.

Data sheets

Data sheets usually provide the saturation characteristics curve. Points should be sampled along the entire region of the curve.

Device data

`Vce`	collector-emitter voltage at the given Ic (in volts, at 25◦?C) at which the saturation characteristics are measured
`Ic`	collector current at the given Vge, in amps, at which the saturation characteristics are measured
`Vge`	gate-emitter voltage, in volts, at which the saturation characteristics are measured

Model

`parameter`	Model description	Default value
`KF`	MOSFET linear region transconductance, in amps (square volt)	1

IGBT: gate charge

The model spec for IGBT gate charge displays the gate charge characteristics at turn-on at the given Vcc and Ic. It shows the gate-emitter voltage, Vge, as a function of gate charge. Usually, the gate charge curve is divided into three distinct regions.

The first region

The first region shows Vge rising at a constant rate until the collector current reaches Ic as a constant gate current is charging the constant gate-emitter capacitance CGS. The total charge supplied to the gate in this region is Qge. This parameter is obtained in data sheets either in the electrical characteristics table or from the gate-charge curve.

The second region

In the second region, Vge is nearly constant as the gate current discharges the internal MOSFET gate-drain capacitance. The charge supplied in this region is Qgc. Like Qge, it is obtained in data sheets either in the electrical characteristics table or from the gate-charge curve.

The third region

In the third region, Vge increases at a constant rate again, as the device is now operating in the linear region. The gate current charges both CGS and the internal MOSFET gate-drain overlap oxide capacitance COXD. Qg and Vg represent a point along the curve in this region. They are obtained either in the electrical characteristics table or from the gate-charge curve. Note that Qg must be greater than the sum of Qge and Qgc. Furthermore, Vg must be greater than the gate-emitter plateau voltage Vge in the second region.

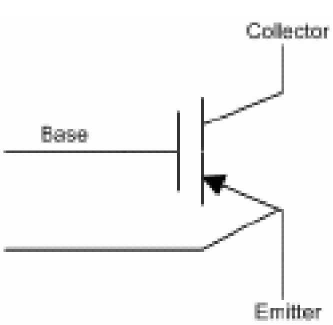
Device data

`Qge`	gate-emitter charge at turn-on at the given Vcc and Ic, in coulombs
`Qgc`	gate-collector charge at turn-on at the given Vcc and Ic, in coulombs
`Qg`	total gate charge at turn-on at the given Vg, Vcc, and Ic, in coulombs
`Vg`	gate voltage at which Qg is measured, in volts
`Vcc`	collector voltage at which Qge, Qgc, and Qg are measured, in volts
`Ic`	collector current at which Qge, Qgc, and Qg are measured, in amps

Model

`parameter`	Model description	Default value
`CGS`	internal MOSFET gate-source capacitance per unit areas, in farads/(square cm)	1.24E-8
`COXD`	internal MOSFET gate-drain overlap oxide capacitance per unit area, in farads/(square cm)	3.5E-8
`AGD`	internal MOSFET gate-drain area, in square m	5E-6

Insulated Gate Bipolar Transistor: Test Node Mapping



Node	Stands for…	In the diagram…
TERM_IC	Collector Current terminal	Current through Collector
TERM_IG	Gate Current terminal	Current through Gate
NODE_VC	Collector Voltage node	Voltage at Collector
NODE_VG	Gate Voltage node	Voltage at Gate
NODE_VS	Source Voltage node	Voltage at Source

Magnetic core spec entry

- Magnetic core: Template-based Model Parameters
- Nonlinear magnetic core: hysteresis curve

Magnetic core: Template-based Model Parameters

Following table lists the template-based model parameters for magnetic core:

Model parameters	Description	Units	Default
AREA	Mean magnetic cross-section	cm^2	1
BM	Saturated flux density	Gauss	1000
BR	Residual flux density	Gauss	1000
GAP	Effective air-gap length (Core gap)	cm	0
HC	Coercive magnetic forcer	Oersted	0.2
ID	Inner diameter (only for torroidal cores)	cm	0
LENGTH	Core path length (only for non-linear ferrite cores)	cm	1

Model parameters	Description	Units	Default
`OD`	Outer diameter (only for toroidal cores)	`cm`	`1`

Nonlinear magnetic core: hysteresis curve

The model spec for nonlinear magnetic cores hysteresis curve estimates the bulk material parameters from the envelope of the B-H curve and the value for the initial permeability of the material. The initial B-H curve starting from the origin is also shown, but it is characterized only by the value for initial permeability.

Data points should be selected from the lower B-H curve in the first quadrant only.

Unlike most models in the Model Editor, in this model the extent of the X-axis plays a role in extracting the model parameters. Owing to the hysteresis (memory) effects in magnetic materials, how the material behaves depends on virtually its entire history. This means the B-H curve will depend on how strong a field it has been subjected to. To accommodate this behavior, the Model Editor simulates a range of fields with magnitudes up to the maximum extent displayed by the X-axis.

Device curve

H	magnetic influence (in Oersteds)
B	magnetic flux (in Gauss)

Conditions

μ_i	initial permeability

Model

parameter	Model description	Default value	Units
A	thermal energy parameter	1E3	amp/meter
AREA	mean magnetic cross-section	0.1	cm2
C	domain flexing parameter	0.2	
GAP	effective air-gap length	0.0	cm
K	domain anisotropy parameter	500	amp/meter
LEVEL	model index	2.0	none
MS	magnetization saturation	1E6	amp/meter
PACK	pack ** (stacking) factor	1.0	none
PATH	mean magnetic path length	1.0	cm
LENGTH	core path length	1.0	cm
OD	outer diameter	1.0	cm
ID	inner diameter	0	cm

Non-linear ferrite family magnetic cores use only LENGTH as user input while toroid family magnetic cores use OD and ID as user input.

Example

If you set the X-axis for a range of -5 to +5, the Model Editor will use fields in that range when extracting the model parameters; the same range of fields will also be used if you set the X-axis for a range of zero to +5. After you have fitted the parameters, you may change the X-axis to see the material's behavior under a different range of external fields.

Note: Numerous evaluations are made when extracting these model parameters. Even the fastest computers will appear to stall momentarily while performing the extraction.

Opamp spec entry

- Opamp: Template-based Model Parameters
- Operational amplifier: large signal swing
- Operational amplifier: open loop gain
- Operational amplifier: open loop phase
- Operational amplifier: maximum output swing
- Operational Amplifier: Test Node Mapping

Opamp: Template-based Model Parameters

Following table lists the template-based model parameters for Opamp:

Model Parameters	Descriptions	Units	Default	Range
`A0`	Open-loop gain	V/V	200K	+
`CINDM`	Diff-mode input cap	F	0	+
`CMRR`	Common-mode reject	V/V	100K	+
`ENW`	Eq. input white noise	V/sqrt(Hz)	10n	+
`GBW`	Gain-BW product	Hz	1 MEG	+
`IB`	Input bias current	A	100p	- +
`IBOS`	Input offset current	A	0	- 0 +
`IBT1`	IB at TMPIB1	A	0	- +
`IBT2`	IB at TMPIB2	A	0	- +
`ISCM`	I short (- source)	A	25m	- +
`ISCP`	I short (+ sink)	A	25m	+
`P0`	GBW excess phase	deg	0	+
`PD`	Quies. power dissip.	W	50m	+

`PSRR`	Power supply reject	V/V	100K	+
`RINDM`	Diff-mode input res	W	10 G	+
`RLOADP`	Load res for VPDIFF	W	2K	+
`ROAC`	AC output res	W	20	+
`ROUT`	DC output res	W	75	>ROA C
`SRM`	Negative slew rate	V/sec	527K	+
`SRP`	Positive slew rate	V/sec	527K	+
`TCIB`	Bias Current TCO	A/C	0	- +
`TCIBOS`	Offset current TCO	A/C	0	- +
`TCVOS`	Offset voltage TCO	V/C	0	- +
`TMPIB1`	Temp measure IBT1	C	0	- +
`TMPIB2`	Temp measure IBT2	C	0	- +
`TMVOS1`	Temp measure VOST1	C	0	- +
`TMVOS2`	Temp measure VOST2	C	0	- +
`VCC`	+ Supply volt	V	15	>VSS
`VMDIFF`	Neg. output diff.	V	1	+
`VOS`	Offset voltage	V	1	- 0 +
`VOST1`	VOS at TMVOS1	V	0	- +
`VOST2`	VOS at TMVOS2	V	0	- +
`VPDIFF`	Pos. output diff.	V	1	+
`VSS`	- Supply volt	V	-15	- +

Note: The `VPDIFF` and `VMDIFF` parameters let you model the opamp output voltage range w.r.t supply rail. If these two are zero, opamp would behave as rail-to-rail opamp.

`VPDIFF` and `VMDIFF` are specified as an algebraic difference from `VCC` and `VSS`, respectively:

```
VPDIFF = VCC-VMAX
```

```
VMDIFF = VMIN-VSS
```

Operational amplifier: large signal swing

The model spec for the opamp large signal swing sets the value of output voltage limiters but also gathers information that will be useful in later screens. The graph shows the largest amplitude output a sinewave signal can be for a given frequency to have no distortion. This is limited by the amplifier's output swing and slew-rate.

Power supply values are the data sheet values used in conjunction with the maximum output values and are not the power supply values for the circuit simulation (which may be different). The opamp model limits the output swing by an amount relative to the power supply, so the output swing limit will track the power supply in the simulation.

About slew-rates: because Model Editor uses primary units (e.g., volts, amps, farads, etc.), the variety of ways of specifying slew-rate needs to be converted to volts/second, e.g., 5V/uS converts to 5,000,000 V/S.

Device data

`+Vpwr`	positive power supply
`-Vpwr`	negative power supply
`+Vout`	maximum positive output swing
`-Vout`	maximum negative output swing
`+SR`	positive-going slew-rate limit
`-SR`	negative-going slew-rate limit
`Pd`	quiescent power dissipation

Macromodel internal parameters

`VC`	output limiter offset (to Vcc)
`VE`	output limiter offset (to Vee)

Operational amplifier: open loop gain

The model spec for opamp open loop gain completes the input stage and inner stage. The compensation capacitor value (Cc) is sometimes available on the data sheet in the circuit diagram of the opamp. If not, then 20-to-30pF is a fair value. For opamps with external compensation, use one of the values on the data sheet for the external capacitor, and use that value for the other input data as well.

Open-loop gain is a ratio of input/output signal, i.e., small-signal amplification. Being a pure number, it has no units. If the gain is specified as 20V/mV, the gain is 20,000; if the gain is specified as 90db, enter 90db (the Model Editor converts xdb to (10^(x/20)).

Unity gain frequency is the intersection of a straight-line extension of the of the mid-band, open-loop, gain roll-off to unity gain (zero decibel). The graph will show gain with only the low-frequency pole included. The high-frequency pole is calculated from open-loop phase margin.

Common-mode rejection ratio (CMRR) has no frequency dependence.

BJT input

Device data

`Cc`	compensation capacitor
`Ib`	input bias current
`Av-dc`	open-loop gain (DC)
`f-0db`	unity gain frequency
`CMRR`	common-mode rejection ratio
`Ibos`	Offset current
`Vos`	Offset voltage

Macromodel internal parameters

`BF`	input transistor beta
`C2`	compensation capacitor
`CEE`	slew-rate limiting capacitor
`GA`	interstage transconductance
`GCM`	common-mode transconductance
`IEE`	input stage current
`RC`	input stage load resistance
`RE`	input stage emitter resistance
`REE`	input stage current source output resistance
`RP`	power dissipation

JFET input

Device data

`Cc`	compensation capacitor
`Ib`	input bias current

`Av-dc`	open-loop gain (DC)
`f-0db`	unity gain frequency
`CMRR`	common-mode rejection ratio
`Ibos`	Offset current
`Vos`	Offset voltage

Macromodel internal parameters

`BETA`	input transistor transconductance
`C2`	compensation capacitor
`CSS`	slew-rate limiting capacitor
`GA`	interstage transconductance
`GCM`	common-mode transconductance
`IS`	input leakage current
`ISS`	input stage current
`RD`	input stage load resistance
`RP`	power dissipation
`RSS`	input stage current source output resistance

Operational amplifier: open loop phase

The model spec for opamp open loop phase adjusts the open-loop unity-gain phase margin, which models the high-frequency pole. Sometimes this value is not available in a table but can be found in a graph. This value is not critical for lower-frequency circuits or lower-Q filters; just use the value we provide, which is typical for normal opamps.

Device data

`Phi`	phase margin (in degrees) @ unity gain frequency

Macromodel internal parameter

`C1`	phase control capacitor

Operational amplifier: maximum output swing

The model spec for opamp maximum output swing adjusts the output drive. The graph shows the maximum output level for a resistive load. The data sheet usually lists an output resistance

`Ro = (Ro_dc) + (Ro_ac)`. Split this value so that (Ro-dc) is about 2 ? (Ro-ac).

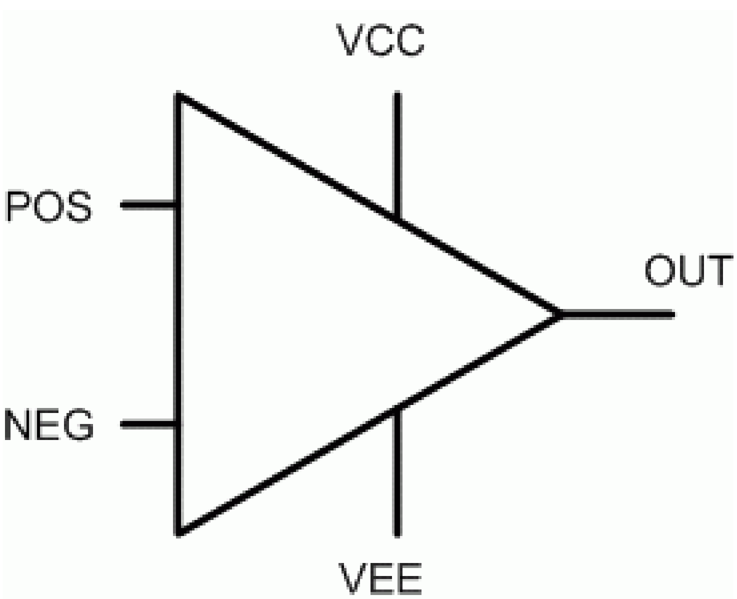
Device data

`Ro-dc`	DC output resistance
`Ro-ac`	AC output resistance
`Ios`	short-circuit output current limit

Macromodel internal parameters

`RO1`	output resistor #1
`RO2`	output resistor #2
`GB`	output stage transconductance

Operational Amplifier: Test Node Mapping



`Node`	Stands for…	In the diagram…
`NODE_POS`	Positive Voltage Source node	Voltage at POS
`NODE_NEG`	Negative Voltage Source node	Voltage at NEG
`NODE_VCC`	Positive Voltage Source node	Voltage at VCC

`NODE_VEE`	Negative Voltage Source node	Voltage at VEE
`NODE_GND`		
`TERM_POS`	Positive current terminal	Current through POS
`TERM_NEG`	Negative current terminal	Current through NEG
`TERM_OUT`	Output current terminal	Current through OUT

Voltage comparator spec entry

- Voltage comparator: transfer function
- Voltage comparator: falling delay
- Voltage comparator: transition time
- Voltage comparator: rising delay

Voltage comparator: transfer function

The model spec for voltage comparator transfer function sets gain values and sets an input offset (to model comparators whose common-mode input includes ground). The model spec shows the transfer function, which is usually not informative, except to tell you that something is happening.

Power supply values are the data sheet values used in conjunction with the maximum output values, but are not the power supply values for the circuit simulation (which may be different).

Device data

`+Vpwr`	positive power supply
`-Vpwr`	negative power supply
`+Vicr`	positive common-mode range
`-Vicr`	negative common-mode range
`Ib`	input bias current
`Avd`	DC gain
`Rl`	output load resistance
`Pd`	power dissipation

Macromodel internal parameters

`BF1`	input stage gain
`BF5`	output stage gain
`RP`	power dissipation resistance
`VI`	input offset

Voltage comparator: falling delay

The model spec for voltage comparator falling delay sets reaction time to input signals. The data sheet usually gives a falling delay, which includes some of the transition in the output waveform (from 100% to 90%). Usually the transition is much faster than the delay and can be ignored (or subtracted from the value). The precise value is not critical, given the unit-to-unit variation.

Device data

`Vst`	input voltage step size
`Vod`	input voltage step overdrive
`td`	delay time

Macromodel internal parameter

`TR3`	input stage reverse transit time

Voltage comparator: transition time

The model spec for voltage comparator transistor time sets the slew rate of the output. The data sheet usually gives a value going from 90% to 10%, which will be within 25% of the full swing time. The precise value is not critical, given the unit-to-unit variation.

Device data

`Vst`	input voltage step size
`Vod`	input voltage step overdrive
`ttr`	transition time

Macromodel internal parameter

`TF5`	output transistor forward transit time

Voltage comparator: rising delay

The model spec for voltage comparator rising delay sets the reaction to input signals, but in the opposite direction of the falling delay. The data sheet usually gives rising delay, which includes some of the transition in the output waveform (from 0% to 10%). Usually the transition is much faster than the delay and can be ignored (or subtracted from the value). The precise value is not critical given the unit-to-unit variation.

Device data

`Vst`	input voltage step size
`Vod`	input voltage step overdrive
`td`	delay time

Macromodel internal parameter

`TR5`	output transistor reverse transit time

Voltage regulator spec entry

- Voltage Regulator: Template-based Model Parameters
- Voltage regulator: reference voltage
- Voltage regulator: adjustment pin current
- Voltage regulator: output impedance
- Voltage regulator: current limit
- Voltage Regulator: Test Node Mapping

Voltage Regulator: Template-based Model Parameters

Following table lists the template-based model parameters for voltage regulator:

Parameter	Description	Unit	Default
`CT`	Timing capacitor	`F`	`1E-15`
`IB`	Quiescent current	`A`	`1E-5`
`IMAX`	Maximum current	`A`	`10`
`IVD1`	Current for dropout	`A`	`1`
`LDREG`	Load regulation	`%/A`	`0.02`
`LINREG`	Line regulation	`%/V`	`1E-5`

M1ILIM	Linear slope of current limit	A/V	1E-3
M2ILIM	Quadratic slope of current limit	A/V^2	0
RT	Timing resistor	Ohm	1E4
TC1IMAX	First temperature coefficient of current limit	/C	0
TC1VD	First temperature coefficient of dropout voltage	/C	0
TC1VR	First temperature coefficient of VR	/C	0
TC2MAX	Second temperature coefficient of current limit	$/C^2$	0
TC2VD	Second temperature coefficient of dropout voltage	$/C^2$	0
TC2VR	Second temperature coefficient of VR	$/C^2$	0
VD1	Dropout voltage at IVD1	V	1.5
VDMIN	Minimum dropout voltage	V	1
VMAIP	Maximum VI-Vo for maximum current	V	0
VR	Output voltage	V	5

Voltage regulator: reference voltage

The model spec for Voltage Comparator Reference Voltage shows the reference voltage across the output (OUT) pin and adjustment (ADJ) pin as the input-output voltage differential increases from 0V to (Vi-Vo)max. The parameter (Vi-Vo)max is used for graphing purposes only. It does not affect the model characteristics.

Device data

Vref	reference voltage

The dropout voltage specifies the minimum input-output voltage differential below, which the circuit ceases to regulate. This parameter is either obtained from the condition of the reference voltage parameter or is given as a parameter by itself in datasheets.

Conditions

`Dropout`	dropout voltage
`(Vi-Vo)max`	maximum input/output voltage differential
`Iomin`	minimum output current to maintain regulation

Model parameters

`VREF`	reference voltage
`N`	emission coefficient

Caution

The minimum output current parameter is obtained from the condition of the reference voltage parameter in datasheets. Ensure that this parameter is also given at the condition Vi-Vo=(Vi-Vo)max.

Voltage regulator: adjustment pin current

The model spec for voltage regulator adjustment pin current displays the adjustment pin current as the input voltage increases from zero volts to (Vi-Vo)max. The parameter (Vi-Vo)max is used for graphing purposes only; it does not affect the model characteristics.

Device data

`Iadj`	adjustment pin current

Model parameter

`BETA`	transconductance of JFET transistor

Example

The adjustment pin current represents an error term in the design equation:

```
Vo = Vref (1 + R2/R1) + (Iadj ? R2)
```

where R1 and R2 are two external resistors. Usually, Iadj is small enough that the voltage (Iadj ? R2) is negligible in the above equation.

Voltage regulator: output impedance

The model spec for voltage regulator output impedance calculates the output impedance over frequency. The output impedance is the impedance seen looking back into the OUT pin, excluding the effects of any external components connected to it. Datasheets usually present the output impedance graphically. Obtain Zout at the same frequency where the ripple rejection value is given. Assume that there is no capacitance connected to the adjustment pin (Cadj=0).

The condition Frequency is used for parameter referencing purposes only. It does not affect the model characteristics.

Device data

`Zout`	low frequency output impedance
`Zero`	dominant zero frequency of output impedance
`RR`	low frequency ripple rejection in decibels

Conditions

`Frequency`	frequency at which Zout and RR are obtained
`IO`	output current at which Zout and RR are obtained

Model parameters

`VAF`	Early voltage of output pass transistor
`CPZ`	output impedance zero capacitor

Voltage regulator: current limit

The model spec for voltage regulator current limit shows the output current as the voltage Vi-Vo increases from zero volts to (Vi-Vo)max as given in the Voltage Reference window when the output is shorted to ground.

The foldback current is the current when the maximum output current is reduced with increasing Vi-Vo voltage. This current is shown as part of the current limit graph given in datasheets. The Model Editor performs a quadratic curve fit on the given data points of the foldback current.

Device data

`Iomax`	maximum output current

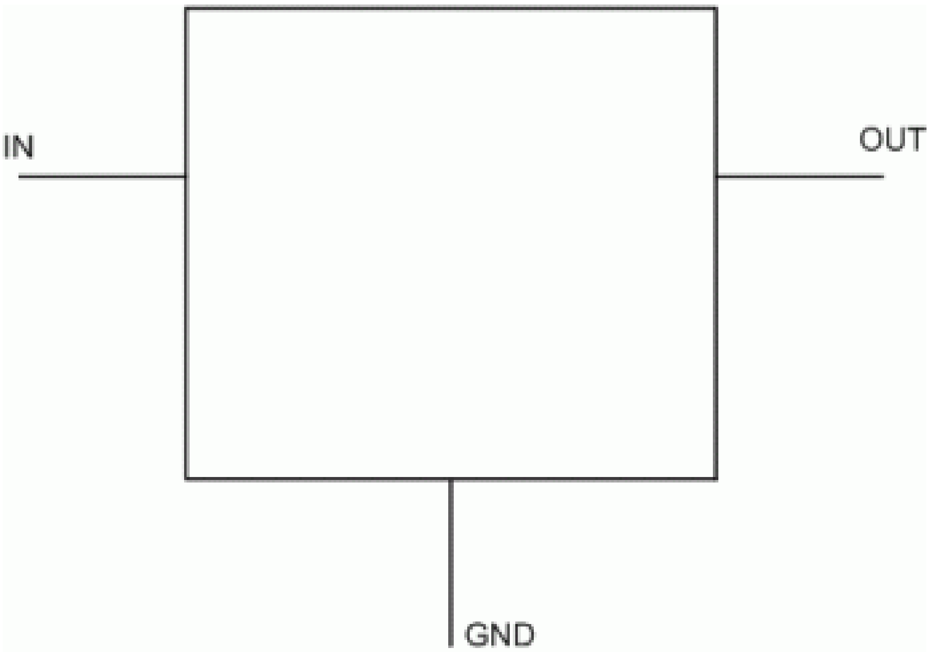
Device curve

`Iofb`	foldback current
`Vi-Vo`	input-output voltage differential

Model parameters

`RB2`	base resistance of output pass transistor
`ESC1`	coefficient of current limit voltage source
`ESC2`	coefficient of current limit voltage source
`EFB1`	coefficient of foldback current voltage source
`EFB2`	coefficient of foldback current voltage source
`EFB3`	coefficient of foldback current voltage source
`EB`	first stage voltage gain

Voltage Regulator: Test Node Mapping



Node	Stands for…	In the diagram…
`NODE_IN`	Input voltage node	Voltage at IN
`NODE_OUT`	Output voltage node	Voltage at OUT
`NODE_GND`	Ground voltage node	Voltage at GND

Voltage reference spec entry

- Voltage reference: reverse dynamic impedance
- Voltage reference: reference voltage
- Voltage reference: temperature drift
- Voltage reference: reverse characteristics
- Voltage reference: forward characteristics

Voltage reference: reverse dynamic impedance

The model spec for voltage reference shows the reverse dynamic impedance with reverse current at a low frequency. The dynamic impedance is the impedance seen looking into the cathode terminal at a given reverse current. Data sheets usually present the reverse dynamic impedance characteristics graphically.

Obtain Rz at nominal temperature. Ir requires positive values.

Device curve

`Ir`	reverse current
`Rz`	dynamic impedance
`Model parameters`	
`NZ`	reverse breakdown coefficient
`RZ`	dynamic impedance

Voltage reference: reference voltage

The model spec for voltage reference reference voltage displays the reverse characteristics as the reverse current increases to the absolute maximum breakdown current. The reverse breakdown voltage is the reference voltage which exhibits tight tolerance and low temperature drift.

Obtain Vref at nominal operating temperature at Ir. Vref requires positive values. The portion of the curve below Vref is shown as an approximation here. It is more accurately modeled in Reverse Characteristics .

Data sheets

Data sheets usually give Irmax under the absolute maximum ratings section.

Device data

`Vref`	reverse breakdown voltage
`Ir`	reverse current at which Vref is obtained
`Irmax`	absolute maximum reverse breakdown current

Model parameters

`RBV`	reverse breakdown reference resistance
`IRMAX`	absolute maximum reverse breakdown current

Voltage reference: temperature drift

The model spec for voltage reference temperature drift shows the reverse breakdown voltage variation with temperature at the reverse current Ir provided in the previous Reference Voltage screen. The curve will pass through Vref at nominal temperature as specified in reference voltage.

The Model Editor does a quadratic curve fit to the given data points. If temperature drift is not shown graphically in datasheets and only the average temperature coefficient is provided, enter data points such that the maximum deviation of Vref divided by the maximum temperature range will result in the correct average temperature coefficient.

Device curve

`Temp`	operating temperature in degrees Celsius
`Vref`	reverse breakdown voltage at Temp

Model parameters

`TC1`	first-order temperature coefficient
`TC2`	second-order temperature coefficient

Voltage reference: reverse characteristics

The model spec for voltage reference reverse characteristics shows the reverse characteristics for reverse voltages up to Vref. See the reference voltage screen for voltages greater than Vref.

Data sheets

Data sheets usually show the reverse characteristics graphically. Obtain data points at nominal temperature. Both Vr and Ir require positive values.

Device curve

`Vr`	reverse voltage
`Ir`	reverse current

Model parameters

`IREV`	reverse saturation current
`NREV`	reverse current coefficient

Voltage reference: forward characteristics

The model spec for voltage reference forward characteristics estimates the parameters IS and RS from three voltage and current values. Try to include data from low current values (where the increase in current is exponential), moderate current values, and high current value (where the increase in current is clearly resistive).

Also, it is sometimes helpful to set up traces for a few values of temperature (use Add Trace) for adjusting XTI.

Device curve

`Ifwd`	forward current @ Vfwd
`Vfwd`	forward voltage across junction for Ifwd

Model parameters

`IS`	saturation current
`N`	emission coefficient
`RS`	series resistance
`IKF`	high-injection "knee" current
`XTI`	IS temperature coefficient

Testing and verifying models created with the Model Editor

Each curve in the Model Editor is defined only by the parameters being adjusted. For example, for the diode, the forward current curve only shows the model of the current equation that is associated with the forward characteristic parameters (such as IS, N, and Rs).

However, the simulator uses the full equation for the diode model, which includes a term involving the reverse characteristic parameters (Such as ISR, NR). These parameters may have a significant effect at low current.

This means that the curve displayed in the Model Editor does not exactly match what is displayed in simulator after a simulation. Make sure to test and verify models using the simulator. If needed, fine-tune the models.

Model Editor Glossary

A

Add trace

Auto Refresh Button

B

BJT

C

Circuit

Component

Create new model button

D

Data sheet information

Design Model

Device

E

Extract button

F

Fixed check box

G

Get model button

Global model

H-I

IGBT

J

JFET

K-L

Library Contents frame

M

.MOD file

Model definition

Model library

MOSFET

N-O

Opamp

Open Library

P

Parameter frame

Part

Q-S

T

U

V

W-X

X Axis Setting

Y

Y Axis Setting

Z

Zoom In

Zoom Out

Add trace

Adds a trace to the plot.

Auto Refresh Button

Enables the automatic redrawing of the graph whenever data is changed.

BJT

Acronym for Bipolar Junction Transistor.

Data sheet information

Data sheet information shows what the manufacturer guarantees for a particular device. The device's operating characteristics will fall within the range specified: a particular device could be near the minimum value of one specification and near the maximum value of another. A typical value is given for some specifications to indicate how most of the devices will operate.

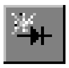
Circuit

A configuration of electrically connected components or devices.

Component

A device or model employed in a circuit to obtain some desired function.

Create new model button



Create a new device model. How to Use <link>

Design Model

A design model applies to one design. You can also create models externally and then manually configure their model files for a specific design.

Device

A simple or complex discrete electronic component, or sometimes a subsystem employed as a unit and therefore thought of as a single component.

Extract button

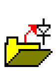
Start the parameter extraction process. Model parameters will be modified according to the specification data. After parameter extraction is finished, all plots will be updated to use the new model parameter values.

Fixed check box

In the Parameters frame, in the Fixed column, select the check box to keep the current value as specified during parameter extraction.

For example, if the current value for EG (activation energy) is 1.110 and you enable Fixed, then EG maintains its value of 1.110 during parameter extraction.

Get model button



Get a model from a model library. How to Use <link>

Global model

A global model is available to any design you create. The Model Editor creates global models by default. You can also create models externally and then manually configure their model files for use in any design.

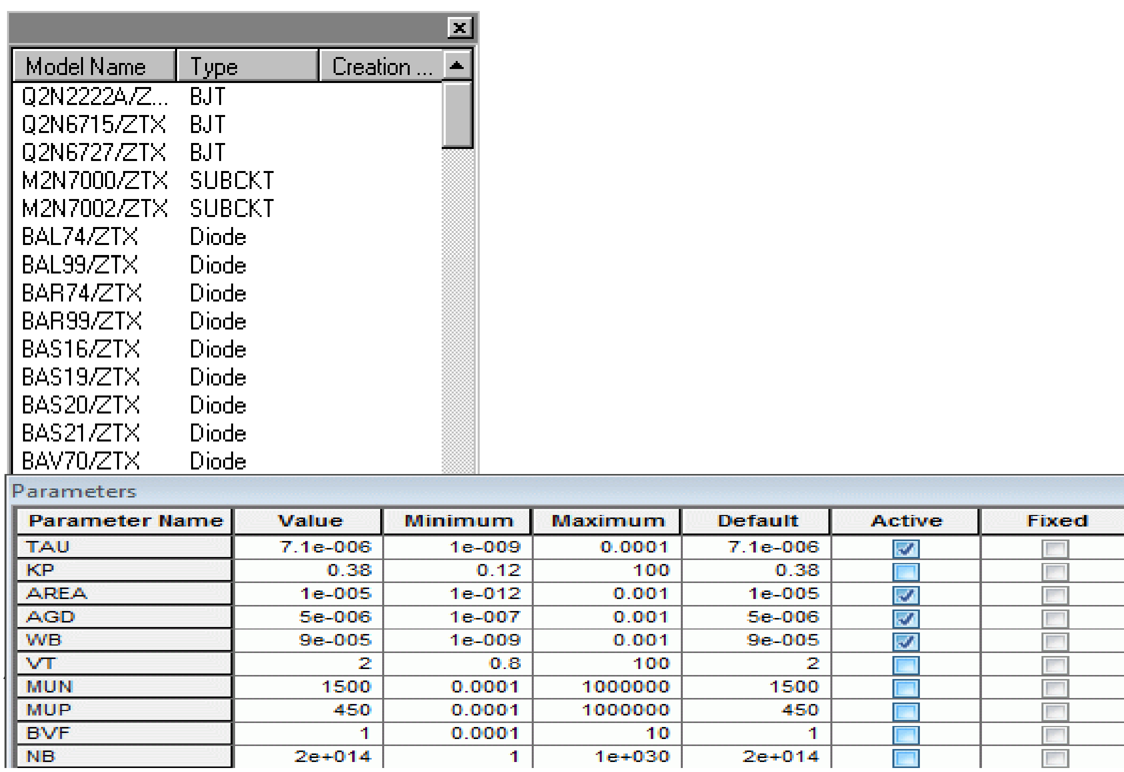
IGBT

Acronym for Insulated Gate Bipolar Transistor.

JFET

Acronym for Junction Field Effect Transistor.

Library Contents frame



of models.

Model definition

An underlying description of the electrical behavior of a part using a set of variable parameters.

Model library

A file that consists of model definitions for devices that you can use in your circuit schematics.

MOSFET

Acronym for Metal Oxide Semiconductor Field Effect Transistor.

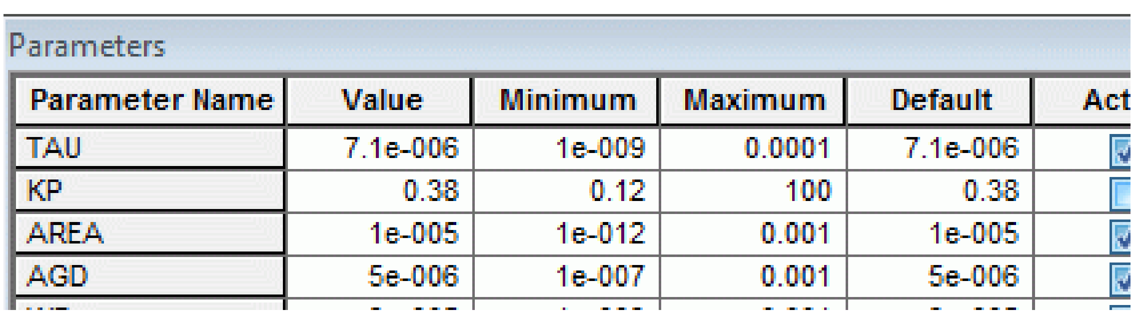
Opamp

Acronym for Operational Amplifier.

Open Library

Open a library in the Model Editor.

Parameter frame


Parameters

Parameter Name	Value	Minimum	Maximum	Default	Act
TAU	7.1e-006	1e-009	0.0001	7.1e-006	☑
KP	0.38	0.12	100	0.38	☐
AREA	1e-005	1e-012	0.001	1e-005	☑
AGD	5e-006	1e-007	0.001	5e-006	☑

Part

The graphical representation of a logical or physical electronic component on the schematic page. A part may have associated attributes.

You can create parts for a specific schematic or extract them from a part library file. Parts can contain nested schematic pages.

Part Definition

The information associated with a graphic part, including the part name and any aliases and attributes, the primitive definition (also called the circuit definition), and the pin assignments. The information is used by other OrCAD applications in the process of generating netlists for simulation or board layout.

Part Library

A file that consists of part definitions for subcircuits or logical or physical components that you can use in your schematic.

Part Library path same as model library

Use this option to save each automatically created part to a part library that has the same name as the model library.

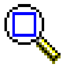
Print

Print any or all of the model information.

Save

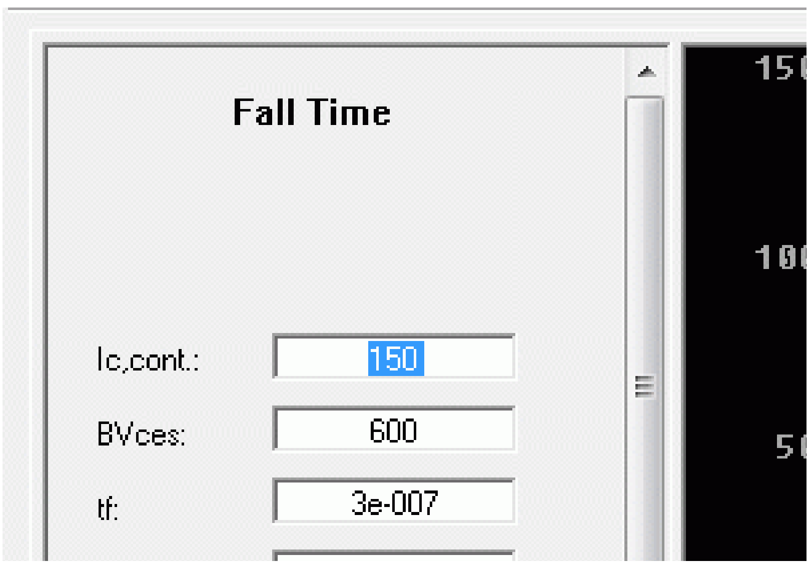
Save the current model to the open library. How to Use <link>

Show All



Set the display to the default view factor and redraw the display.

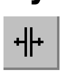
Spec Entry frame



Subcircuit

A small collection of components working together to deliver a function.

Sync Splitters button



The Sync Splitters button enables the synchronization of the split bars in the Spec Entry frame. This button does not correspond to any menu commands.

Trace

A trace can be a digital waveform, an analog waveform, or a bus made up of a maximum of 32 digital signals.

Update Graph

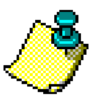
The Update Graph button redraws the device graphs after you add or edit values in the spec. This is the same as choosing Redraw from the View menu or pressing F5.

Use default base part set

Use this option to base the automatically created parts on OrCAD's default parts.

User-defined part library

Use this option to save all the automatically created parts to a specific file. Enter this file name in the Part Library Name text box.



The Model Editor will continue to save parts for newly created models in this part library, even in future sessions.

Use user-defined base part set

Use this option to base the automatically created parts on the parts contained in a custom part library. Enter this file name in the Part Library with Base Parts text box.

Vcomp

Acronym for Voltage Comparator.

View Area

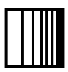
Zoom into the area you specify with the mouse by clicking and dragging the mouse to form a view window. You can specify the area before or after you select View Area.

View Stack

A view stack is a memory queue of display views.

There is one view stack per plot. View changes for all Y axes for a plot go into the view stack for that plot.

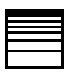
X Axis Setting



Toggle the X-axis scale between Linear and Logarithmic.

You cannot put either axis into a log scale if either end of the axis range is zero or negative.

Y Axis Setting



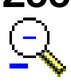
Toggle the Y-axis scale between Linear and Logarithmic.

You cannot put either axis into a log scale if either end of the axis range is zero or negative.

Zoom In

Zooms in by a factor of 2 around the point you specify with the mouse.

Zoom Out



Zooms out by a factor of 2 around the point you specify with the mouse. The View Fit limits will not be exceeded. If the View Fit limits are exceeded in either the X or Y direction, then only the direction in which they are not exceeded will zoom out by the full factor of 2.