IC-CAP 2010.08
July 2010
Introduction and Basics

This is the default Notice page

Acknowledgments

Errata The IC-CAP product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof." To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Introduction

The IC-CAP Modeling System is part of Agilent EEsof EDA's high-frequency electronic design automation (EDA) solutions. Agilent EEsof EDA offers a full array of design tools that streamline and strengthen the engineering process. These tools include powerful system and circuit simulators, 2.5-D and 3-D electromagnetic simulators, device modeling systems, and physical design tools.

The IC-CAP Modeling System

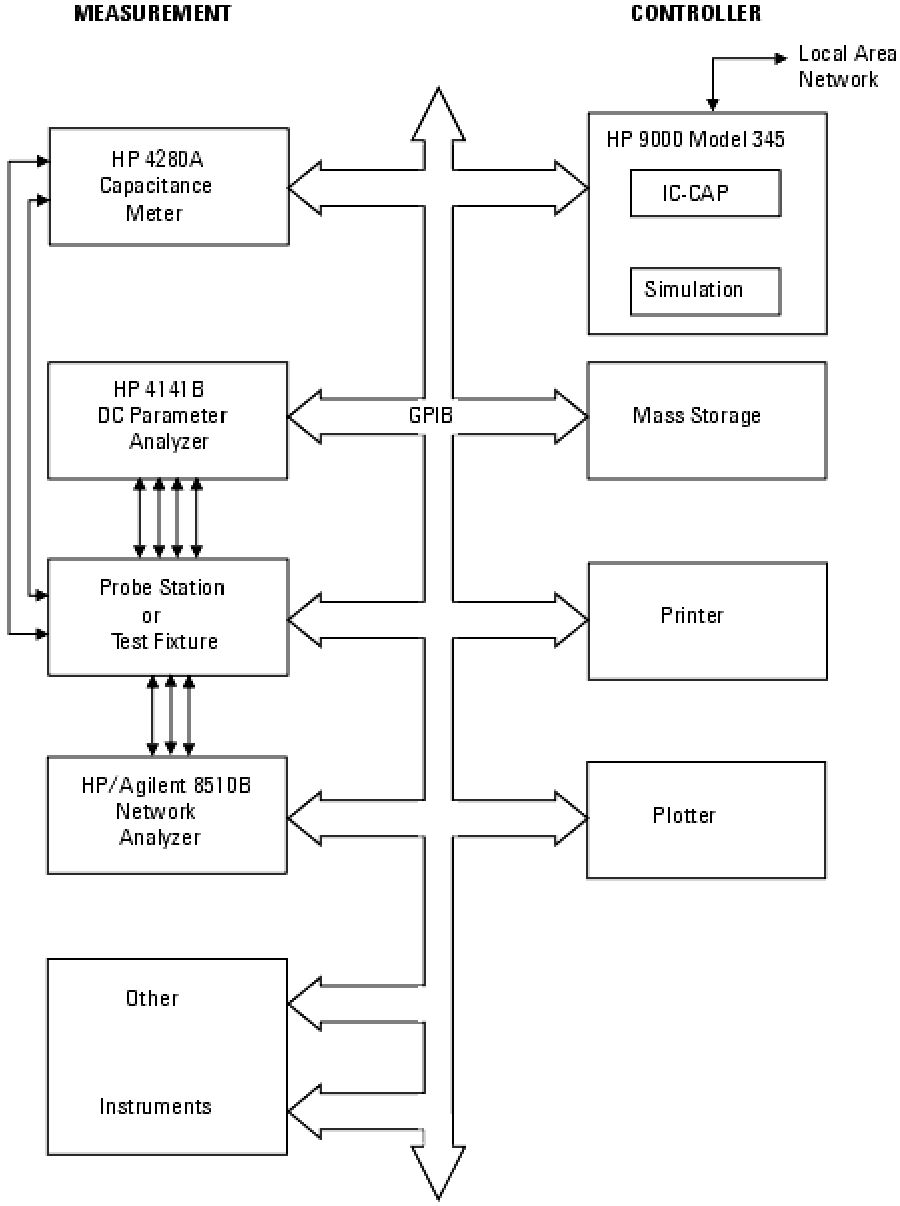
The IC-CAP Modeling System is used to measure semiconductor device and circuit modeling characteristics and analyze the resulting data.

To use IC-CAP, you will require the following:

- A workstation
- Instruments that perform DC, Capacitance, AC, and Time-Domain measurements
- A test fixture
- A test device
- The IC-CAP software

The following figure shows the IC-CAP global configuration.

IC-CAP Global Configuration



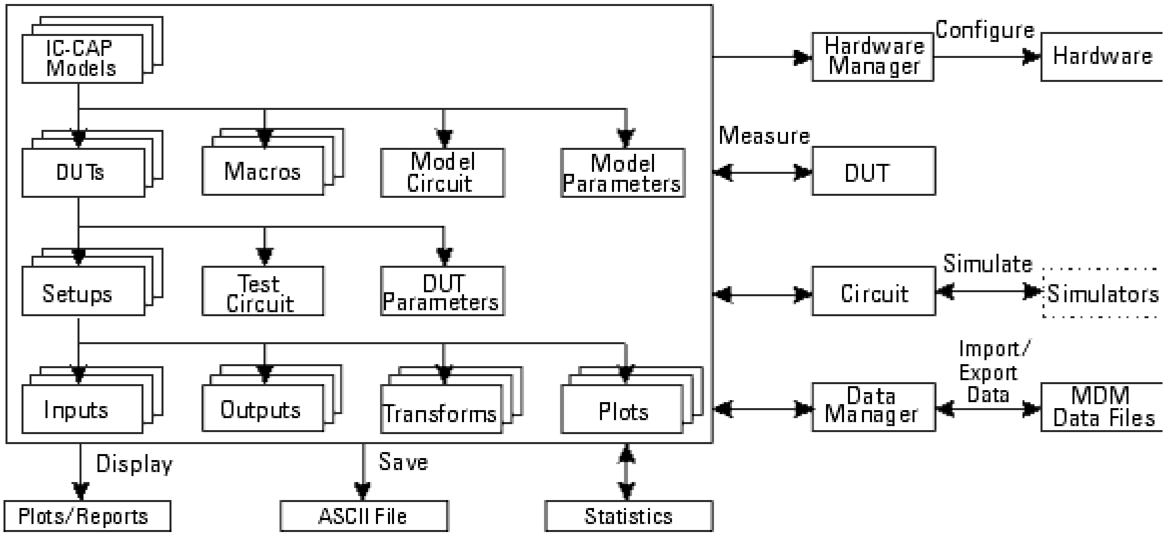
The IC-CAP System Architecture

The IC-CAP system centers around a data storage area containing device and circuit characteristics. Initially, these characteristics are obtained from measurements or simulations. After gathering your data, you can perform a variety of operations on the data, including mathematical transformation, extraction, optimization, analysis, or archival to a file or a data base. The analysis features available in IC-CAP allow detailed study of the characteristics and the data can be saved for future use.

Mathematical transformations are used to extend the available set of characteristics, while extractions and optimizations provide feedback to the model in order to obtain agreement between measured and simulated data.

The following figure illustrates the IC-CAP system architecture.

IC-CAP System Architecture



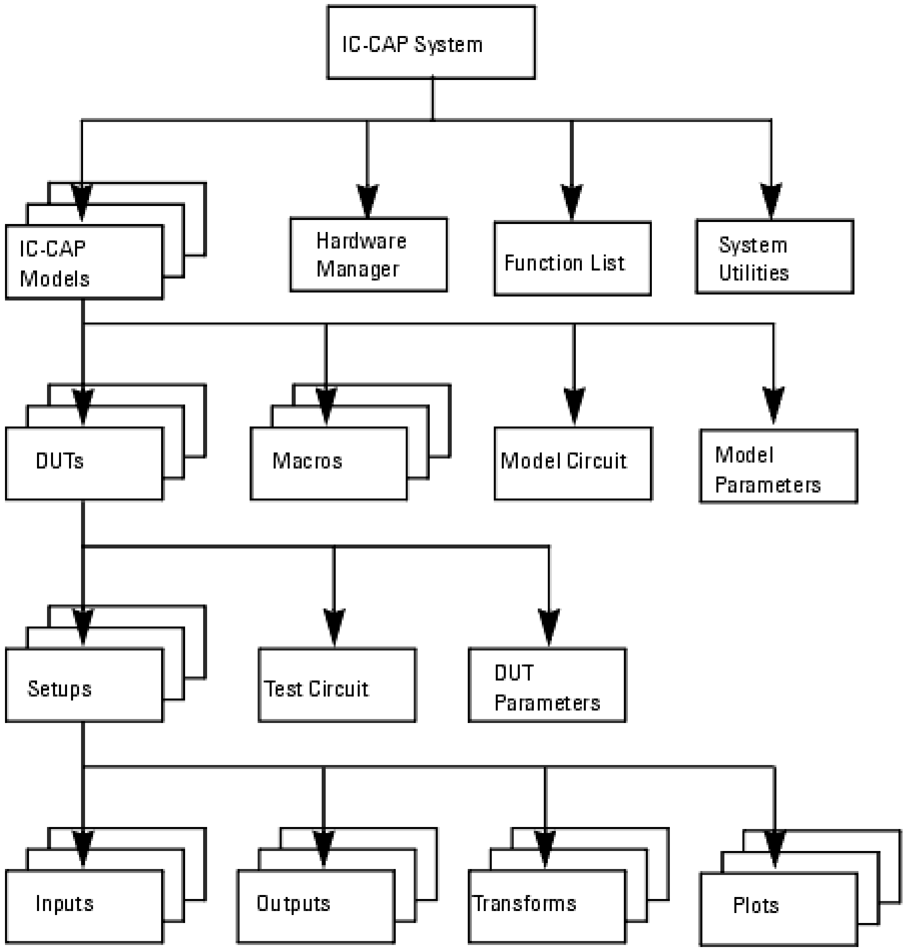
System Functional Areas

The four main functional areas of the IC-CAP system are:

- Modeling
- Hardware Management
- Function List
- System Utilities and Operations

The following figure shows the general organization of the IC-CAP system.

General Organization of the IC-CAP System



Modeling is the key functional area. You use this function to:

- Test devices and circuits using predefined models
- Modify existing models by editing
- Create new models by describing their physical and electrical characteristics
- Create macros to automate measurement and extraction processes
- Create the DUTs contained in a model by describing their physical and electrical characteristics
- Define test circuits and modify parameters for individual DUTs
- Create Setups for a particular DUT by defining:
    - The connection of a device or circuit to the test instruments
    - The inputs and outputs of the device or circuit
    - The transformations of the outputs used to calculate parameter values or additional data
    - The characteristics of the plots used to display the results of your tests

The hardware management function is used to create and modify the GPIB configuration of the test instruments. The function list is used to view the available characterization functions. These functions include mathematical transformations, model parameter extractions, and special analysis functions, such as optimization. System utilities are used to create and modify the global configuration. Operations include simulator selection and directory specification.

General Operating Procedures

IC-CAP provides a complete set of procedures for characterizing devices and circuits. Each of these procedures can be executed from easily accessed menus or programmed into IC-CAP macros. The system can be used for routine operations with a minimum of training. The general procedure for characterizing a device consists of the steps described here and shown in the following figure.

Installation

Install the device in a test fixture. The instruments must be connected to the computer and IC-CAP via the GPIB bus.

Load an Existing Model or Create a New Model

Load an existing model from the Model List after starting the program or create a new model by modifying an existing model or by starting with a blank template.

Measure or Create Device Characteristics

Execute IC-CAP's *Measure* command to measure or create device characteristics. The program takes control of the measuring instruments and executes their functions. The instruments connected to the test fixture generate source signals at the input nodes of the test device. Then the response signals are recorded by the instruments connected to the output nodes of the device. The responses are recorded by the instruments and the measured data are loaded into the IC-CAP database.

Extract Model Parameter Data from the Measured Data

Execute IC-CAP's *Extract* command to calculate the parameters that control the electrical behavior of the device from the measured output data.

Simulate

Send the circuit definition and the extracted parameters to the simulator. The simulator generates a simulated data set for comparison with the measured data set.

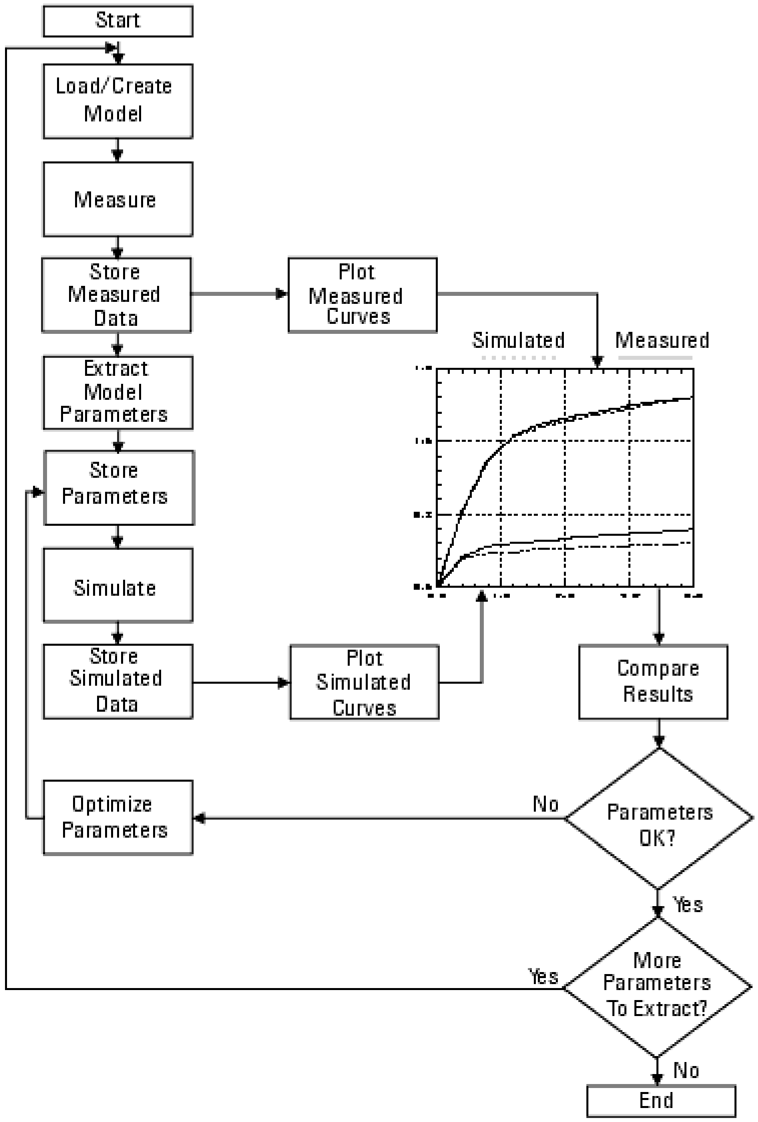
Optimize

Optimize the parameters to achieve the best possible fit between measured and simulated data.

Results

Display or print results in graphic or tabular reports.

General Functionality of the IC-CAP System



IC-CAP Terminology

The following table lists the terms used throughout the documentation to describe the basic parts of the IC-CAP system.

Term	Description
Model	Complete instructions for characterizing a device, including a description of the device, measured and simulated data, and functions that are performed on the data.
Model Parameters	Parameters that govern the behavior of the device, and whose values are common to all devices being measured for a particular characterization.
DUT (Device Under Test)	A particular configuration of the device being measured and the hardware being used to perform the measurements.
Test Circuit	Additional circuitry that is connected between the device being measured and the hardware being used on a particular DUT.
DUT Parameters	Parameters that govern the behavior of the device, and whose values may be different for each DUT.
Circuit	A description of the device, in SPICE deck format.
Setup	Instructions for performing a measurement or simulation on a DUT. A setup contains definitions of stimuli, responses, mathematical transformations, extractions, and graphical or tabular reports.
Measurement	The application of a set of stimuli to an actual device to obtain a set of responses that are used in the characterization process.
Input	The definition of a stimulus in a measurement or simulation. Input includes the array of data that results from this definition.
Output	The definition of a response in a measurement or simulation. Output includes the arrays of data that result from performing those operations.
Extraction	The calculation of the parameters that control the device behavior, typically from measured data, but sometimes from simulated data or data sheet values.
Transform	The definition of a function that is performed on IC-CAP data to create a new set of data or calculate parameter values. Transform includes the arrays of data that result from performing the function.
Simulation	The calculation of a set of data from a mathematical model of the device.
Optimization	The iterative adjustment of parameter values in order to achieve the best possible agreement between measured and simulated data.
Plot	Instructions for generating and displaying IC-CAP data, either in graphical or tabular format.
Macro	A group of IC-CAP functions combined into a single operation.

Graphic User Interface - Menu Descriptions

See *Menu Descriptions* (intro)

Miscellaneous Information

See *Miscellaneous* (intro)

Using the IC-CAP Interface

Main Window

File Menu

New	Creates a blank, untitled model file and displays a symbol for it in the work area of the Main window. Highlight the text below the symbol and type the desired name for the new model. Double-click to open the Model window.
Open	Enables you to open an existing model file.
Examples	Enables you to quickly open a model file in the examples directory
Edit	Opens the selected Model window (select the model symbol, then the command).
Auto Execute	Runs the AutoExecute macro in the selected model if it is declared or opens the selected Model window if AutoExecute in not declared.
Save As	Enables you to save one or more of the models currently in memory.
Change Directory	Enables you to change the directory to which data will be saved during the current session.
<File Names>	Contains list of most recently loaded files.
Exit	Exits IC-CAP. The Save As dialog box appears enabling you to save any models currently in memory before exiting.

Edit Menu

Undo	Undoes the most recent Cut, Copy, or Paste command.
Cut	Cuts the selected model to a buffer from which it can be pasted. This enables you to load and view another model by the same name-without overwriting the first one-and choose between them, pasting the one you cut if desired.
Copy	Copies the selected model to a buffer from which it can be pasted. By copying and pasting, you can have one or more copies of a given model open at the same time.
Paste	Pastes the contents of the buffer. When pasting a model you cut, the same name is retained. When pasting a model you copied, an underscore (_) character is appended to the name, as well as a numerical indicator, which is automatically incremented.
Delete (No Undo)	Deletes the selected model from the work area.

Tools Menu

System Variables	Displays the System Variables window for setting values for global (system) variables. You can create user-defined variables by typing names and values here. You can also type names and values of supplied variables here, or click the **System Variables** button and use the dialog box to select variables, view descriptions, and set values. Note that the names of supplied variables are reserved and cannot be used for variables you create.
Print	The Print button enables you to save the currently displayed variables table to file. You are prompted for a filename. The file is saved to the current work directory with a default .asc extension.
File	• Open-Enables you to open a previously saved Variables Table file (.vat). Change directories as needed, select the desired file, and click **OK**. • Save As-Enables you to save a variables table to file. Change directories as needed, type the desired name in the Selection field, and click **OK**. • Close-Closes the variables window.
Edit	• Undo-Undoes the previous copy or paste • Copy-Copies the currently defined global variables enabling you to paste them in the Model window at the Model, DUT, or Setup level. • Paste-Enables you to paste variables copied at the Model, DUT, or Setup level to the global level.
System GUI Items	Displays the System GUI Items window. For details, see *Creating Graphic User Interfaces* (prog).
Simulation Debugger	Displays the Simulation Debugger Window.
Stop Simulator	Stops any currently running piped simulator. The command is most useful when using the ADS simulator with the Root model to ensure correct simulations can be made after changing directories to test other models, and to allow model regeneration.
License Status	Displays a window with dynamically updated license information: • The codewords currently in use • The codewords currently available To release a specific license, select it and click **Release**.
Hardware Setup	Displays the Hardware Setup Window.
Statistics	Launches the Statistics program, if licensed.
Select Simulator	Enables you to change the default simulator after startup. Note: this selection is overriden by a SIMULATOR variable. For detailis, refer to *Selecting a Simulator* (sim).
Functions	Displays the Function Browser dialog box for reviewing available functions.
Plot Options	Displays the Plot Options dialog box, which enables you to define trace options, plot options, and text annotation.
Options	
File Debug	Toggles the debugging facility on and off. When on, messages are recorded in the file .icdebug.
Screen Debug	Toggles the debugging facility on and off. When on, messages are displayed in the IC-CAP Status window.
View GUI Pages	If toggled on, the GUI Items page will be shown in Model windows when the window is first opened.
Status Window to Top	If toggled on, the Status window pops to the front of the screen anytime new messages are displayed in it.
Diagnostics	Executes internal diagnostics.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Hardware Setup Window

The Instrument Setup window enables you to define interface filenames, active instruments, instrument addresses, and unit names to be used in Setups.This window can also be used to control a particular instrument manually.

File Menu

The File menu allows you to perform basic file management commands as well as exit the Hardware Setup window.

Open	Enables you to load an instrument configuration from a previously saved file.
Save	Enables you to save the current instrument configuration to a file of your choosing (the default filename is .icconfig).
Close Window	Closes the Hardware Setup window.

Tools Menu

This set of commands provides basic GPIB capabilities to communicate with the instruments on the bus. These may be useful for debugging an instrument driver or manually setting an instrument to a certain state that is not supported by IC-CAP. The Status panel displays continuously updated information about GPIB activity.

The Tools menu offers the following choices:

- Interface
- Address
- Send/Receive
- Settings
- Macros
- Serial Poll

Interface

Status	Provides the current GPIB bus status.
Change	Closes the current interface file and opens a new one.
Lock	Provides exclusive access to the bus.
Unlock	Releases the I-O Lock on the bus.
Reset	Resets all the instruments on the bus using the Interface Clear command.

Address

Set	Sets the address to which certain commands apply. Some of these commands are: • Send String • Receive String • Serial Poll • Address > Listen • Address > Talk • Address > Check NOTE: Use the command Address > Set first to set a target GPIB address on which to perform various communications such as "Send String" or "Receive String."
Who Are You?	Polls all addresses other than those at which the CPU is configured. Provides a list of all instruments attached to the bus and powered on, along with their addresses.
Check	Polls active address for any response.
Listen	Sets the instrument at the active address to the Listen state.
Talk	Sets the instrument at the active address to the Talker state.

Send/Receive

Send String	Prompts for a string that will be sent to the instrument at the active address. Carriage return and line feed characters can be included using "\r" and "\n", respectively.
Receive String	Sets CPU to Listen status and the active address to talk status, and collects data from the active address until an EOI is received. Usually used with Send String. The result is also placed in the HPIB_READ_STRING system variable if you have defined it.
Receive PEL String	Same as Receive String, but skips the Talk/Listen setup which should be done using the Send Byte command. The result is also placed in the HPIB_READ_STRING system variable if you have defined it.
Display String	Displays data most recently read by the CPU via a Read String operation.
Send/Read/Display	Lets you send a command to the instrument at the active address and receive a response from the instrument. The response is displayed in the Status panel. The result is also placed in the HPIB_READ_STRING system variable if you have defined it.
Send Byte	Sends one command byte specified as a decimal integer. For example, use 63 to send UNLISTEN.

Settings

Timeout	Specifies the timeout value in seconds. Specifying 0 (zero) disables the use of timeout, which is not recommended.

Macros

Specify	Prompts you for the name of an GPIB Analyzer macro file.
Execute	Executes commands contained in the GPIB Analyzer macro file. For the syntax of commands, refer to *GPIB Analyzer* (meas).

Serial Poll

Polls the active address and displays the status byte in decimal form.

Instrument Menu

The Instruments menu offers a variety of instrument operations to assist you in making measurements.

Find	Queries the GPIB bus for instruments.
Display	Displays a list of found instruments on the GPIB bus.
Usage	Displays a list of currently active instruments.
Zero Sources	Forces zero to all the found instruments.
Self Test	Causes the execution of self tests on instruments currently connected to the system, powered up, and listed in the Instrument List.

View Menu

The View menu allows you to toggle on and off the screen debugger and the toolbar.

Screen Debug	Enables you to toggle on and off the low-level debugging facility that produces detailed debug messages on each GPIB transaction.
Toolbar	Enables you to toggle the toolbar on and off.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Model Window

File Menu

Open	Enables you to open parts of a model file previously saved with the Save As command in the Model window (see *Opening Parts of a Model File* (intro)).
Save As	Enables you to save parts of a model file (see *Opening Parts of a Model File* (intro)).
Printer Setup	Opens the Print Setup dialog box (see *Printing and Plotting* (intro)).
Import Data	
Active Setup	Enables you to import an ASCII-based MDM data file, of a pre-determined format, to the selected setup.
All Setups in Active DUT	Enables you to import an ASCII-based MDM data file, of a pre-determined format, to all setups in the selected DUT.
All DUTs in Model	Enables you to import an ASCII-based MDM data file, of a pre-determined format, to all setups in all DUTs of the current model.
Export Data	
Active Setup	Enables you to export an ASCII-based MDM (.mdm) or Dataset (.ds) data file, of a pre-determined format representing the selected setup.
All Setups in Active DUT	Enables you to export an ASCII-based MDM (.mdm) or Dataset (.ds) data file, of a pre-determined format representing all setups in the active DUT.
All DUTs in Model	Enables you to export an ASCII-based MDM (.mdm) or Dataset (.ds) data file, of a pre-determined format representing all DUTs in the model.
Extracted Deck	Enables you to export the circuit block in a SPICE deck format.
Write Model MDIF	Writes the Parameters table to a file in MDIF format, which can be read by ADS or Series IV.
Close	Closes the Model window, but the model file remains in memory. To re-open a model, double-click the icon in the Main window.

Related Topics:

Introduction (intro)

Edit Menu

Undo	Undoes the last Cut, Copy, or Paste action.
Text	Applies to text in the model parts DUTs, Circuit, and Macros.
Cut	Cuts selected text.
Copy	Copies selected text.
Paste	Pastes copied or cut text to other textual model parts.
Find	Enables you to find text.
Replace	Enables you to find and replace text.
Goto Line	Enables you to highlight selected line number.
Cut Setup	Enables you to cut a variety of model parts for pasting within other parts of that same model or to another model.
Copy Setup	Enables you to copy a variety of model parts for pasting within other parts of that same model or to another model.
Paste	Enables you to paste a variety of model parts you cut or copied. You can paste within the model that you cut or copied from or to another model.
Delete Setup (No Undo)	Enables you to delete a variety of model parts.

Measure Menu

Active Setup	Performs a measurement for the active setup.
Active DUT	Performs a measurement for all setups in the active DUT

Extract Menu

Active Setup	Performs all extraction transforms for the active setup.
Active DUT	Performs all extraction transforms for all setups in the active DUT.

Simulate Menu

Active Setup	Simulates the active setup.
Active DUT	Simulates all setups in the active DUT.

Optimize Menu

Active Setup	Performs all optimization transforms for the active setup
Active DUT	Performs all optimization transforms for all setups in the active DUT

Data Menu

Plots	
Display All	
In Active Setup	Displays all currently defined plots for the active setup.
In Active DUT	Displays all currently defined plots for the active DUT.
in Model	Displays all currently defined plots for the current model.
Close All	
In Active Setup	Closes all displayed plots for the active setup.
In Active DUT	Closes all displayed plots for the active DUT.
in Model	Closes all displayed plots for the current model.
Clear Data	
Measured	Clears all measured data *In Active Setup* , *In Active DUT* , or *In Model* .
Simulated	Clears all simulated data *In Active Setup* , *In Active DUT* , or *In Model* .
Both	Clears both measured and simulated data *In Active Setup* , *In Active DUT* , or *In Model* .

Tools Menu

Simulation Debugger	Displays the Simulation Debugger Window.
Stop Simulator	Stops any currently running piped simulator. The command is most useful when using the ADS simulator with the Root model to ensure correct simulations can be made after changing directories to test other models, and to allow model regeneration.
Organize Model	Displays a window that enables you to add, delete, reorder, and rename DUTs, Macros, and Variables in the model. Selecting the Organize DUT button on the window displays a window that enables you to add, delete, reorder, and rename Setups and Variables. Selecting the Organize Setup button on that window displays a window that enables you to add, delete, reorder, and rename Inputs, Outputs, Transforms, Plots, and Variables.
Refresh Last Dataset...	Re-exports the current data to the last exported dataset (.ds).
Plot Optimizer	Displays the Plot Optimizer window.
Plot Options	Displays the Plot Options dialog box, which enables you to define trace options, plot options, and text annotation.
Hardware Setup	Displays the Hardware Setup Window.

Macros Menu

Execute	Enables you to execute any macro available in the model regardless of the visible folder.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Plot Window

File Menu

Print	Opens the Print dialog box which enables you to print to the specified printer, a file, or copy to the Windows clipboard.
Save Image...	Enables you to save the current plot configuration to a file of your choosing.
Printer Setup...	Opens the Print Setup dialog box (see *Printing and Plotting* (intro)).
Close	Closes the Plot window.

Options Menu

Replot	Refreshes the plot.
Update Annotation	Updates the plots annotation.
Edit Definition...	Displays the Plot Editor.
View Data...	Displays the plot's data file.
Copy to Clipboard	Copies a plot image to the Windows clipboard.
Rescale	Zooms in to the selected area of a plot.
Set scale	Sets the Manual rescale dialog box to the plot's current scaling values.
Autoscale	Turns the autoscale mode on or off. When turned on, the graph automatically rescales as data changes. The status appears in the graph's title.
Manual rescale	Displays the Manual rescale dialog box which enables you to fully describe all three axes of XY plots in terms of minimum value, maximum value, number of major divisions, and number of minor divisions.
Draw Diag Line	Draws a diagonal line connecting two clicked points and its slope, with both X and Y axis intercepts. If you do not click two points first, erases the diagonal line.
Copy to Variables	Copies the X and Y values of a rescale rectangle to system variables. Four variables (X_LOW, X_HIGH, Y_LOW, and Y_HIGH) are reserved for this purpose.
Error	
Show Relative Error	Toggles the MAX and RMS relative errors in the footer area on or off.
Show Absolute Error	Toggles the MAX and RMS absolute errors in the footer area on or off.
Select Whole Plot	All the points in the measured/simulated datasets will be used to calculate the error.
Select Error Region	Identifies the selected region as the error region. A green box that delimits the error calculation replaces the white box.
Plot Options	Displays the Plot Options dialog box, which enables you to define trace options, plot options, and text annotation.
Session Settings	
Area Tools	Toggles the graph's area tools on or off.
Legend	Toggles the graph's legend on or off.
Text Annotation	Toggles the graph's text annotation on or off.
Title	Toggles the graph's title on or off.
Header	Toggles the header on or off.
Footer	Toggles the footer on or off.
Exchange Black-White	Reverses the black and white settings for the graph's grid, text, and background.
Color	Toggles color on or off for the traces and markers. When *Color* is off, the traces and markers are the same color as the graph's grid and text.
Reset to Saved Options	Resets the current session settings back to the saved Plot Options. This menu pick is not available from Scatter, Histogram, or CDF plots that were opened by the Statistic Package.
Save Current Settings	Saves the current session's settings as the saved Plot Options. This menu pick is not available from Scatter, Histogram, or CDF plots that were opened by the Statistic Package.

Optimizer Menu

Open Optimizer...	Opens the Plot Optimizer window.
Enable/Disable Plot	Enables or disables the Plot window. Enabling the plot window synchronizes it with the Plot Optimizer window.
Global Region	
Reset	Deletes all existing global trace regions on the plot and defines a new global trace region.
Add	Adds a global trace region without deleting existing global trace regions.
Delete All	Deletes all global trace regions.
Autoconfigure and Enable	Automatically enables and configures the inputs in a Plot window.
Disable All Traces	Disables all inputs in a Plot window.
trace	
Set as Both Target and Simulated	Configures the selected trace with its measured data set as Target and its simulated data set as Simulated.
Set as Target vs.	Configures the selected trace as Target.
Set as Simulated vs.	Configures the selected trace as Simulated.
Disable	Disables the selected trace.
Trace Optimizer Region	
Reset	Deletes all existing trace optimizer regions for the selected trace and defines a new trace optimizer region.
Add	Adds a trace optimizer region for the selected trace without deleting existing trace optimizer regions.
Delete All	Deletes all trace optimizer regions for the selected trace.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Multiplot Window

File Menu

Print	Opens the Print dialog box which enables you to print to the specified printer, a file, or copy to the Windows clipboard.
Save Image...	Enables you to save the current plot configuration to a file of your choosing.
Printer Setup...	Opens the Print Setup dialog box (see *Printing and Plotting* (intro)).
Close	Closes the Plot window.

Options Menu

Replot	Refreshes the plot.
Update Annotation	Updates the plots annotation.
Copy to Clipboard	Copies a plot image to the Windows clipboard.
Autoscale	Turns the autoscale mode on or off. When turned on, the graph automatically rescales as data changes. The status appears in the graph's title.
Error	
Show Relative Error	Toggles the MAX and RMS relative errors in the footer area on or off.
Show Absolute Error	Toggles the MAX and RMS absolute errors in the footer area on or off.
Select Whole Plot	All the points in the measured/simulated datasets will be used to calculate the error.
Select Error Region	Identifies the selected region as the error region. A green box that delimits the error calculation replaces the white box.
Plot Options	Displays the Plot Options dialog box, which enables you to define trace options, plot options, and text annotation.
Session Settings	
Area Tools	Toggles the graph's area tools on or off.
Legend	Toggles the graph's legend on or off.
Text Annotation	Toggles the graph's text annotation on or off.
Title	Toggles the graph's title on or off.
Header	Toggles the header on or off.
Footer	Toggles the footer on or off.
Exchange Black-White	Reverses the black and white settings for the graph's grid, text, and background.
Color	Toggles color on or off for the traces and markers. When *Color* is off, the traces and markers are the same color as the graph's grid and text.
Reset to Saved Options	Resets the current session settings back to the saved Plot Options.
Save Current Settings	Saves the current session's settings as the saved Plot Options.

Optimizer Menu

Open Optimizer...	Opens the Plot Optimizer window.
Enable/Disable Plot	Enables or disables the Plot window. Enabling the plot window synchronizes it with the Plot Optimizer window.
Global Region	
Reset	Deletes all existing global trace regions on the plot and defines a new global trace region.
Add	Adds a global trace region without deleting existing global trace regions.
Delete All	Deletes all global trace regions.
Autoconfigure and Enable	Automatically enables and configures the inputs in a Plot window.
Disable All Traces	Disables all inputs in a Plot window.
trace	
Set as Both Target and Simulated	Configures the selected trace with its measured data set as Target and its simulated data set as Simulated.
Set as Target vs.	Configures the selected trace as Target.
Set as Simulated vs.	Configures the selected trace as Simulated.
Disable	Disables the selected trace.
Trace Optimizer Region	
Reset	Deletes all existing trace optimizer regions for the selected trace and defines a new trace optimizer region.
Add	Adds a trace optimizer region for the selected trace without deleting existing trace optimizer regions.
Delete All	Deletes all trace optimizer regions for the selected trace.

Plots Menu

Select Plot	Displays a list of available plots. The plot you choose becomes the selected plot.
Unselect All	Unselects the currently selected plot.
Zoom Plot	Displays a list of available plots. The plot you choose is displayed in zoom format.
Full Page Plot	Displays a list of available plots. The plot you choose is displayed in full page format.
Undo Zoom	Resets the Multiplot window to the default format.
Selected Plot Menu	
Reset to Saved Options	Resets the Plot Options to the last saved options.
Plot Options	Displays the Plot Options dialog box, which enables you to define trace options, plot options, and text annotation.
Optimizer	
Open Optimizer	Opens the Plot Optimizer window.
Enable/Disable Plot	Enables or disables the Plot window. Enabling the plot window synchronizes it with the Plot Optimizer window.
Global Region	
Reset	Deletes all existing global trace regions on the plot and defines a new global trace region.
Add	Adds a global trace region without deleting existing global trace regions.
Delete All	Deletes all global trace regions.
Autoconfigure and	Automatically enables and configures the inputs in a Plot window.

Enable	
Disable All Traces	Disables all inputs in a Plot window.
trace	
Set as Both Target and Simulated	Configures the selected trace with its measured data set as Target and its simulated data set as Simulated.
Set as Target vs.	Configures the selected trace as Target.
Set as Simulated vs.	Configures the selected trace as Simulated.
Disable	Disables the selected trace.
Trace Optimizer Region	
Reset	Deletes all existing trace optimizer regions for the selected trace and defines a new trace optimizer region.
Add	Adds a trace optimizer region for the selected trace without deleting existing trace optimizer regions.
Delete All	Deletes all trace optimizer regions for the selected trace.
Scaling	
Replot	Refreshes the plot.
Rescale	Zooms in to the selected area of a plot.
Set scale	Sets the Manual rescale dialog box to the plot's current scaling values.
Autoscale	Turns the autoscale mode on or off. When turned on, the graph automatically rescales as data changes. The status appears in the graph's title.
Manual rescale	Displays the Manual rescale dialog box which enables you to fully describe all three axes of XY plots in terms of minimum value, maximum value, number of major divisions, and number of minor divisions.
Graphic	Updates the plots annotation.
Draw Diag Line	Draws a diagonal line connecting two clicked points and its slope, with both X and Y axis intercepts. If you do not click two points first, erases the diagonal line.
Copy to Variables	Copies the X and Y values of a rescale rectangle to system variables. Four variables (X_LOW, X_HIGH, Y_LOW, and Y_HIGH) are reserved for this purpose.
Area Tools	Toggles the graph's area tools on or off.
Legend	Toggles the graph's legend on or off.
Text Annotation	Toggles the graph's text annotation on or off.
Title	Toggles the graph's title on or off.
Header	Toggles the header on or off.
Footer	Toggles the footer on or off.
Error	
Show Relative Error	Toggles the MAX and RMS relative errors in the footer area on or off.
Show Absolute Error	Toggles the MAX and RMS absolute errors in the footer area on or off.
Select Whole Plot	All the points in the measured/simulated datasets will be used to calculate the error.

Select Error Region	Identifies the selected region as the error region. A green box that delimits the error calculation replaces the white box.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Plot Optimizer Window

File Menu

Open...	Enables you to open an existing plot optimizer file.
Save As...	Enables you to save the current plot optimizer configuration to a file of your choosing.
Clear Plot Optimizer	Disables all traces and regions in the plots, disables all open plots, and clears the Parameters table.
Reset Option Table	Resets all options in the Options table to the default values.
Close	Closes the Plot Optimizer window.

Plots Menu

Enable All	Enables all open Plot windows in a model file.
Disable All	Disables all open Plot windows in a model file.

Simulate

Simulate All	Simulates all enabled traces.

Optimize

Run Optimization	Performs an optimization of the enabled traces.
Tune Fast...	Opens a tuner window which enables you to adjust the parameter values. You can see the effects in a plot as you vary parameter values using the tuner. Recalculates constantly as a slider is moved. While the tuner window is open, you can change the *Min* and *Max* values.
Tune Slow...	Opens a tuner window which enables you to adjust the parameter values. Recalculates only when a slider is released. You can see the effects in a plot as you vary parameter values using the tuner. While the tuner window is open, you can change the *Min* and *Max* values. It's best to use Tune Slow when the time per calculation is slow (about one second or more).

Tools

Store Parameters	Stores values in the Parameters table for later recall.
Recall Parameters	Recalls the values stored in the Parameters table.
Undo Optim	After running an optimization, restores the parameters to their previous state.
AutoSet Min and Max	Automatically sets the parameter values.
Reset Min and Max	Restores the parameter values to their default values.

Windows

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Simulation Debugger Window

The Simulation Debugger window displays the circuit input deck and the results of the simulation in text form. To display data, open the Simulation Debugger window and then run a simulation (either from the DUT or Setup level). The data is displayed in the Input, Output, and Command File regions, however, the Command File region is used only with the Saber simulator and displays the Saber commands generated from the IC-CAP Setup.

For debugging, edit the data in the Input editor, then run a simulation from the debugger to test the changes. To run a simulation from the debugger, use the Manual Simulation command on the File Menu. If the changes are successful, change the circuit definition to reflect your changes.

Note
After running a simulation using the SPICE3 simulator, the Output region of the debugger displays the message ".print card ignored since rawfile was produced." To display the output of the simulation in text form, issue the Manual Simulation command.

File Menu

Manual Simulation	Executes a simulation using the circuit deck in the Input field of the debugger. The Debugger Input field is not linked to the Model Circuit Definition editor or Input and Output setups, so edits made in the field have no effect on IC-CAP and are irrelevant to a simulation run from a DUT or Setup. To make changes permanent, make them in the Model Circuit Definition editor.
Save Input File	Provides a dialog box for specifying a filename for saving the contents of the Input editor to a file.
Save Command File	Provides a dialog box for specifying a filename for saving the contents of the Command editor to a file.
Save Output File	Provides a dialog box for specifying a filename for saving the contents of the Output editor to a file.
Close	Closes the Simulation Debugger window.

Windows Menu

The Windows menu provides a quick method of bringing a different window to the foreground. All currently open windows, including those that are minimized, are listed on the menu. Individual models are listed on the Model window submenu. Choose the desired window and it is displayed in front of all other IC-CAP windows.

Miscellaneous

Clearing Data from Memory

The Clear command allows you to clear from memory the data for the current setup. You can clear measured data, simulated data, or both. This command is useful when you have already measured all setups of a DUT, or all DUTs that make up the model, and need to make a change to one setup. You can clear the measured data for that setup and re-measure that setup.

Tuning Parameters

By including the TUNER statement in a PEL macro, you can tune parameters during macro execution. To tune parameters once the dialog box appears:

1. Position the pointer over the slider associated with the parameter you want to tune.
2. Press the left mouse button and drag the slider in the desired direction. Notice the plot is dynamically updated as you change values.
3. To close the dialog box and return control to the macro, click **OK**; to abort the macro, click **Cancel**.

Cut and Copy Selections

When you cut and copy model parts at the DUT and Setup levels, you are prompted to choose between the DUT or Setup (depending on which is active) and a model part available in the active folder. For example, in the Measure/Simulate folder, you can choose to cut or copy the Setup itself or selected Inputs and/or Outputs. Likewise, in the Test Circuit folder, you can choose to cut or copy the DUT itself or the Test Circuit.

- Select the appropriate option and click **OK**.

Adding an Interface Name

To add an interface name to the list, click **Add Interface**. In the dialog box that appears, supply the interface filename. An interface card by that name should have been previously installed. When you choose **OK**, the existence of a card by that name is verified.

For information about installing and configuring an interface card, see *Check the Supported Instrument Interfaces for Windows installations* (winstall) or *Check the Supported Instrument Interfaces for UNIX installations* (uinstall).

Deleting an Interface Name

To delete an interface filename from the list, select the interface name and click **Delete Interface**.

Creating the List of Active Instruments

The Instrument List potentially displays all instruments attached to the GPIB bus. Once on the list, you can configure each instrument to set the address and assign unit names. You can create this list automatically or manually.

- To create the Instrument List automatically:

  Click **Rebuild**. This clears the current Instrument List, polls all available GPIB addresses. and creates a complete list of instruments connected to the GPIB bus, providing they are powered up.

- To Create the Instrument List Manually:

  Select the instrument name and click **Add to List**. This enables you to create a list of instruments that are not necessarily currently powered up or connected to the bus.

Deleting an Instrument from the Instrument List

To delete an individual instrument from the list, select it and click **Delete**. You can delete an instrument any time, regardless of its power status.

> **Note**
> To save instrument options before deleting instruments from the list, with the Instrument Options folder active, choose **File > Save As**. Select **Instrument Options (.iot)**, set the path and filename as desired, and click **OK**.

Deleting All Instruments from the List

To delete all instruments from the Instrument List, click **Delete All**. A confirmation dialog box appears to confirm this operation because all instrument options will also be removed.

> **Note**
> To save instrument options before deleting instruments from the list, with the Instrument Options folder active, choose **File > Save As**. Select **Instrument Options (.iot)**, set the path and filename as desired, and click **OK**.

Configuring an Instrument

To define the GPIB bus address of an instrument in the Instrument List, as well as assign unit names, select that instrument and click **Configure**.

Interface

Reflects the names of any interface files added in the Hardware Setup window. Select one to change its address and assign units.

Instrument Address

Enables you to set the address of an instrument on the GPIB.

Unit Table

Contains default unit names for an instrument. Edit these names as needed.

> **Note**
> After assigning unit names here, specify those same names for the inputs and outputs (in the Measure/Simulate folder) as needed.

Program Basics

This section covers the basic concepts and skills you need in order to work successfully with the program.

To install the program, follow the instructions in *Windows Installation* (winstall) or *UNIX Installation* (uinstall).

Documentation Conventions

The IC-CAP documentation uses consistent visual cues, standard text formats, and special terminology so that you can locate and interpret information easily. The following table lists these documentation conventions.

Documentation Conventions

Type style	**Used for**
Italic	New terms, directory names, filenames.
ALL CAPITALS	Acronyms.
Initial Capitals	Names of keys on your keyboard (such as, Esc, Back Space, or Return), menu items, command names, dialog names and options.
Bold	Menu names, command names, items from a list, filenames or project names that you select when following a procedure.

Making Selections with the Mouse

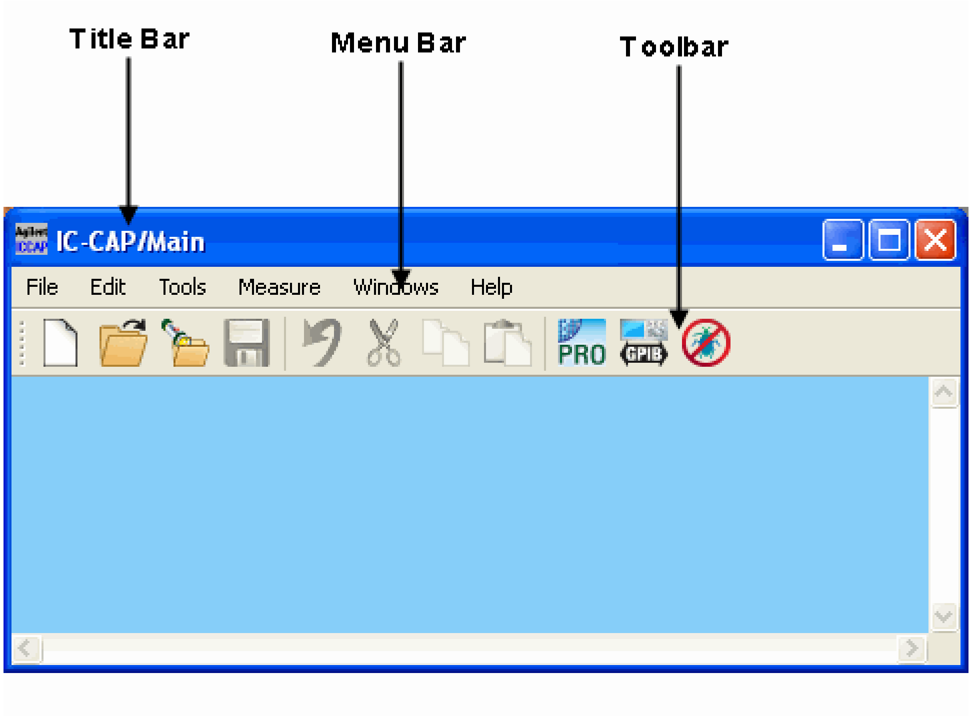
The mouse is used to make selections and to open and close program windows.

- To select an item, move the mouse pointer to the item, then click (press and release) the left mouse button.
- To select a command from a menu:
  - Move the mouse pointer to the command and double-click (two clicks in rapid succession) the left mouse button.
  - Move the mouse pointer to the command and press, but do not release, the left mouse button. Drag (slide) the pointer to the command and then release.
- To cancel a menu, click anywhere outside the menu. (Or, you can press F10 or Esc.)
- To display the x- and y-axis values of a particular point on a data trace, move the mouse pointer to the point and click the middle mouse button.
- To display a list of available choices for a field, move the mouse pointer over the list pointer at the right end of the field, then click the left mouse button.

Working with Windows

A window contains an application, such as a model, or an information dialog box. The following figure shows the parts of the IC-CAP Main window.

An Example IC-CAP Window



You can move, resize, and arrange windows for best usage of the screen. For details on window management, refer to your operating system documentation.

Working with Menus and Commands

You carry out an action by choosing a command. Program commands are listed on menus on the window menu bar. The following table lists menu conventions.

Menu Conventions

Menu Convention	Description
Dimmed command name	The command is not available now.
Ellipsis points (...) displayed after a command name	Opens a dialog box for completing information needed to carry out the command.
A triangle (>) at the right side of the menu command	Opens a sub menu for the command.

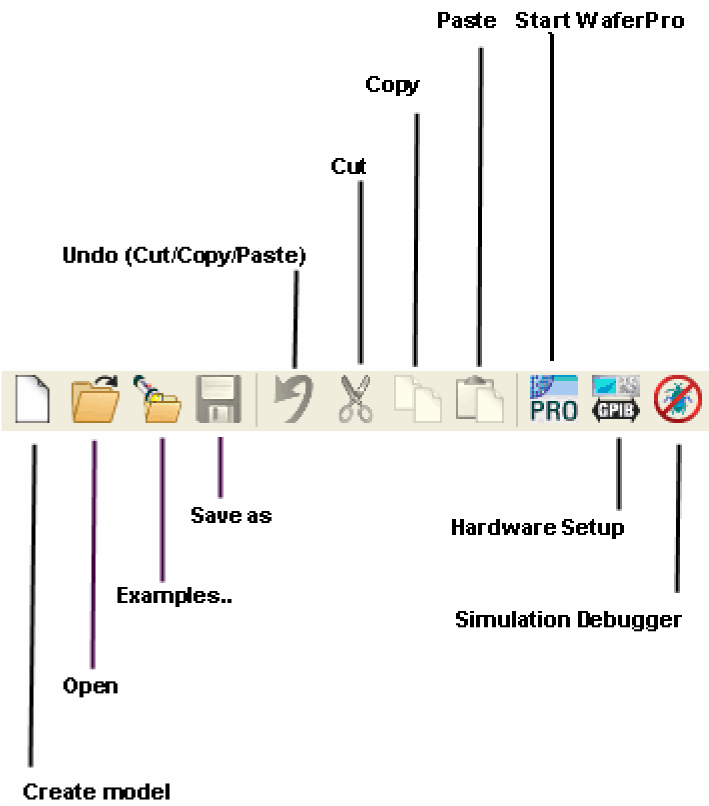
Using the Tearoff Menu

Pull-down menus can be separated from the menu bar and placed anywhere on your desktop. To tear off a menu, click the dashed line at the top of the menu and drag the menu to a new desktop location.

Using the Toolbar

Frequently-used commands are available as buttons on the toolbar in each design window. When you move the mouse pointer over a button, a balloon displays a label identifying the function of that button. The following figure shows the toolbar in the IC-CAP Main window.

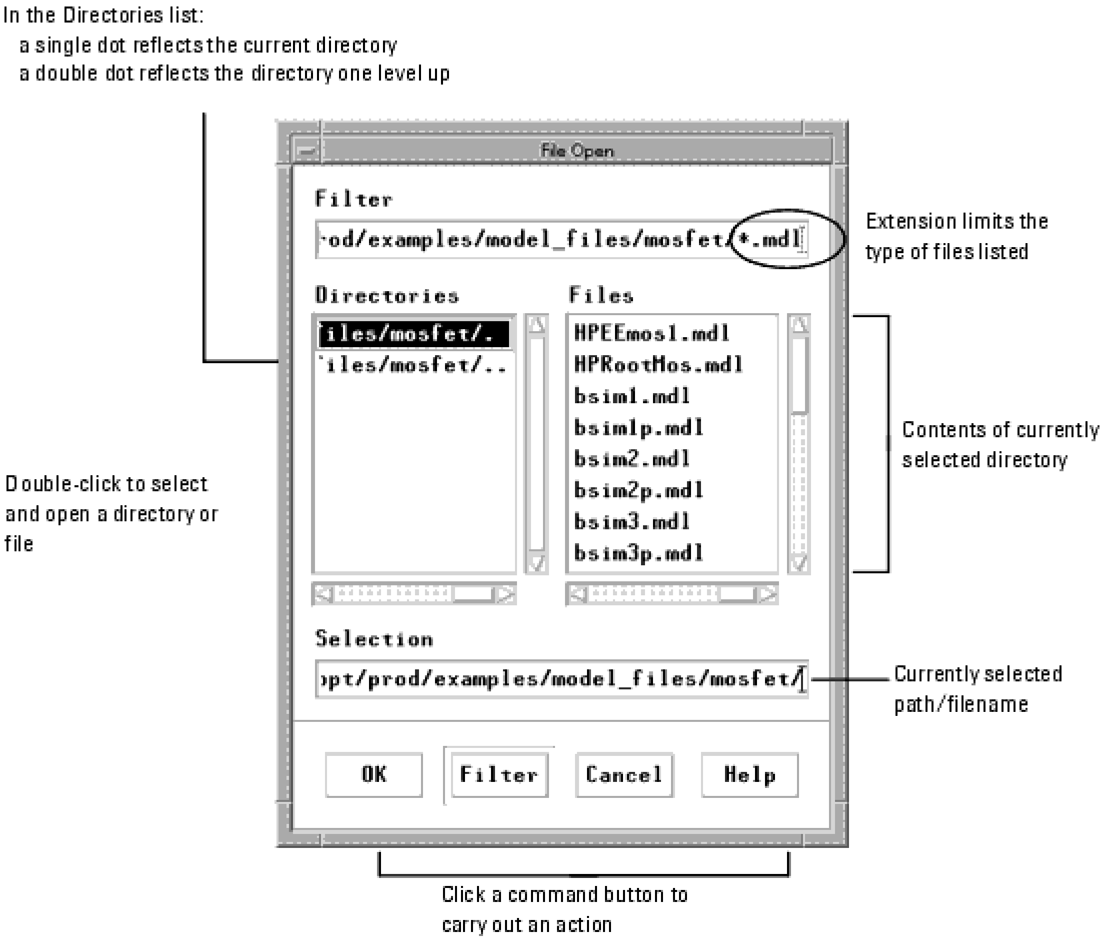
IC-CAP Main Window Toolbar



Working with Dialog Boxes

The program uses a dialog box to display messages or to request information needed to carry out a command. The following figure shows an example dialog box.

An Example Dialog Box



Providing Dialog Box Information

List Box

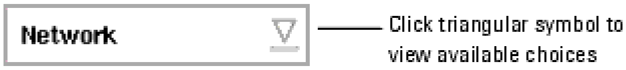
Contains the available choices. If there are more choices than can fit in the list box, you can use the scroll bars to access more choices. Click the item you want to select.

Text Box

Enter the required information in the fields or edit information currently entered. One field has the keyboard focus (where keystrokes appear). This field is identified by a highlighted border and a blinking input cursor that looks like an I-beam.

Drop-down List

Opens after you click the triangular symbol at the end of a bar-like button. Click the item you want to select.



Command and Option Buttons

Initiate immediate action when selected. By default, the choice that has a darker border is active. To choose the default, press **Enter** or click **OK**.

To implement the selections made in a dialog box:

- To make changes in a dialog box, provide the information required, then choose **OK** to complete the command and close the dialog box.
- To make changes, but keep the dialog box open, choose **Apply**.
- To close a dialog box without completing the command, choose **Cancel**.

Prompts

A prompt dialog box requires information to carry out the current command. You must make a choice and close the dialog box before you can take any other action in the program.

Error Messages

An error dialog box opens after an operation in which one or more errors have occurred. All errors resulting from a particular operation are displayed in one box. You must clear the error dialog box before you can continue with IC-CAP operations. All error messages generated during a particular session are written to a file in your home directory named *.icerrlog*. The .icerrlog file is cleared each time IC-CAP is run.

Table and Text Editors

A table editor is used to display and edit groups of information.

The *keyboard focus* field is identified by a highlighted border and a blinking input cursor. Normally the input cursor looks like an I-beam. To change the location of the keyboard focus or reposition the input cursor within a field, click in the field.

The number of visible rows in a list or table is specified by system variables. Changing a window size does not affect a list or table size. Refer to the section on *System Variables* for details on changing variables.

The following table lists keystrokes that you can use while editing fields in a table or text editor. Some keystrokes may not be available on all keyboards. IC-CAP Editor Select and Copy Mouse Operations lists the mouse operations used to select and copy text. All selections described are primary selections unless noted otherwise.

Table and Text Editor Keystrokes

Keystroke	Action
Right Arrow	Move forward one character
Left Arrow	Move back one character
Down Arrow	Move to next line or field
Up Arrow	Move to previous line or field
Home	Move to beginning of line
End	Move to end of line
Backspace	Delete previous character
Shift Backspace	Delete previous word
Delete	Delete selected text
Shift Delete Char	Delete next word
Tab	Move to next field
Return	Begin a new line

Using the Pop up Menu as a Shortcut

A pop-up menu, available in table and text editors, enables access to many common commands with a minimum of mouse movement.

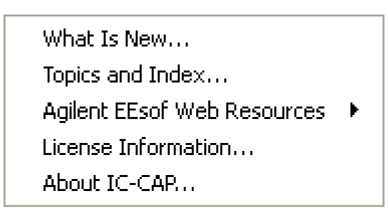
IC-CAP Editor Select and Copy Mouse Operations

Operation	Action
Click left button	Move insertion and destination cursors
Double click left button	Select word
Triple click left button	Select line
Quad click left button	Select entire editor
Drag left button	Select to end of drag
Shift click left button	Extend selection
Shift drag left button	Extend selection to end of drag
Click middle button	Copy selection to new location
Drag middle button	Select (secondary) to end of drag and copy to destination cursor

Operations for copying and pasting text to and from terminal windows are platform-dependent. For details, refer to your workstation's documentation.

Using Help

Help messages are available for each IC-CAP window. Window level help is displayed when you click *Help* in the top menu bar of a specific window.



From the Help menu, you can access this information:

What Is New...	Displays important differences between the PC and UNIX version of IC-CAP.
Topics and Index...	Displays the main contents for the online documentation. Clicking a listed section opens the file for that section.
Agilent EEsof Web Resources	Displays telephone, fax, and e-mail numbers for accessing Agilent EEsof Worldwide Customer Support.
License Information	Displays the environment/license variables and machine information, all licenses found in the license.lic file, all license servers serving Agilent EEsof licenses on your network, and the current status of all installed licenses.
About IC-CAP...	Displays version and copyright information.

You can display a Help message for a specific dialog box by clicking the Help button in the dialog box.

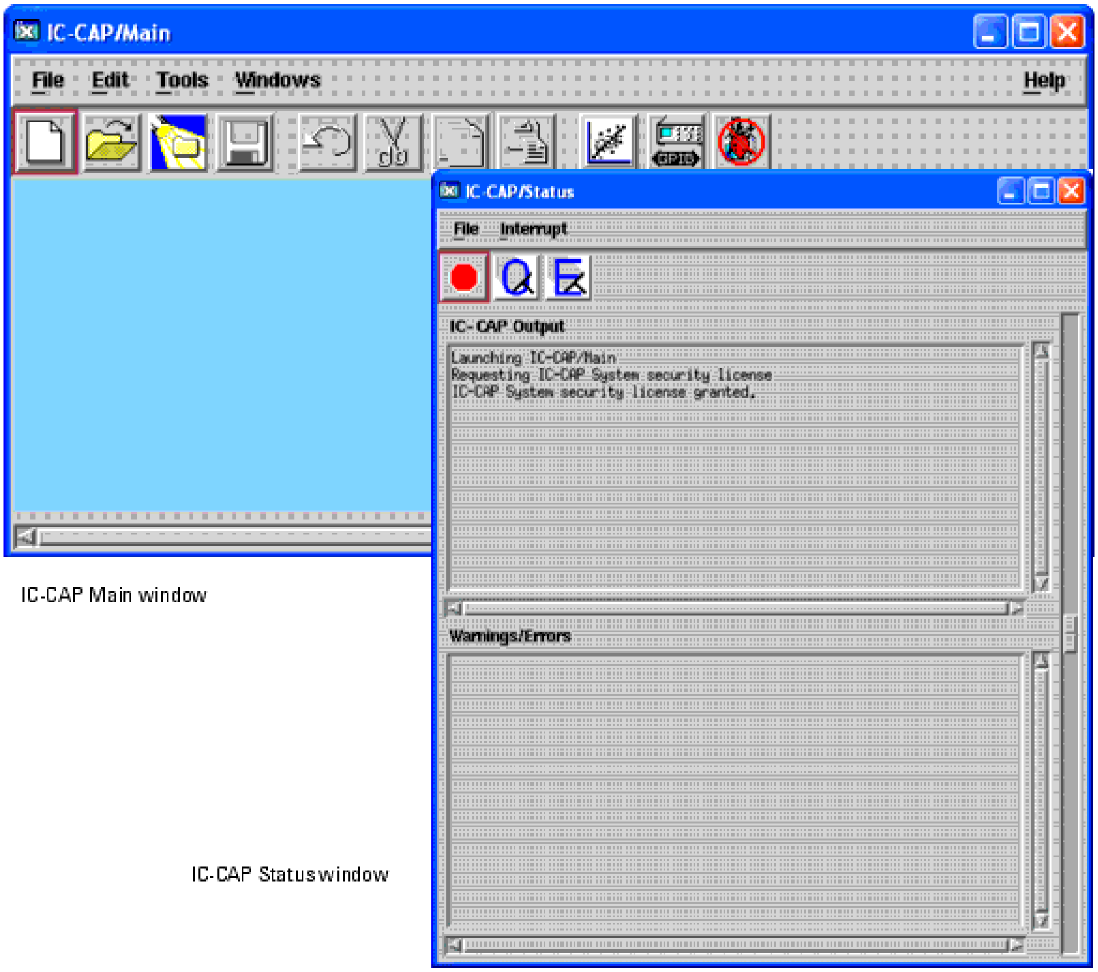
Starting the Program

You can start IC-CAP from any directory. However, after you have saved modified model files, you may find that starting the program from the directory that contains your model files simplifies archiving operations.

To start the program:

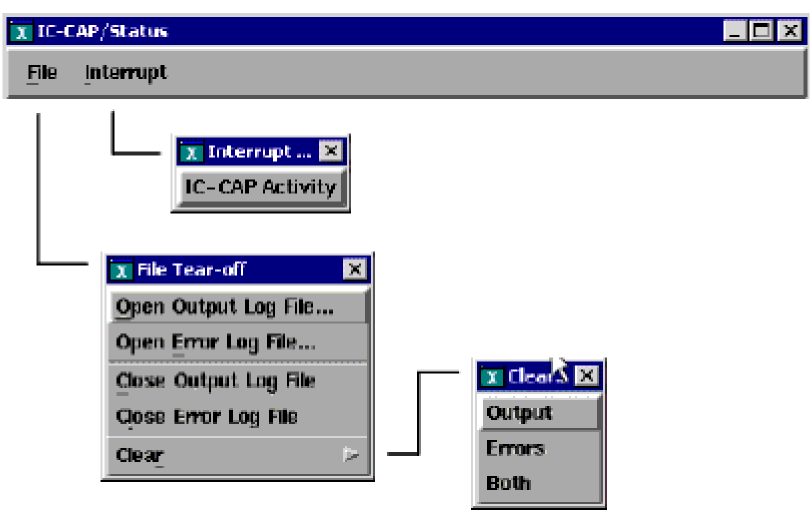
1. Change to the working directory.
2. At the system prompt, type `iccap` and press Enter.

The IC-CAP Main window and the IC-CAP Status window open.



The IC-CAP/Status window displays messages about the program activity and Warnings/Errors messages. In the IC-CAP/Status window, you can open and close output and error log files, and interrupt a process. The icons allow you to interrupt a process (the red stop sign), clear the output log files (the one with the "O"), and clear the error log files (the one with the "E"). The slider along the Status window's right side enables you to adjust the relative sizes of the Output and Warnings/Errors panes.

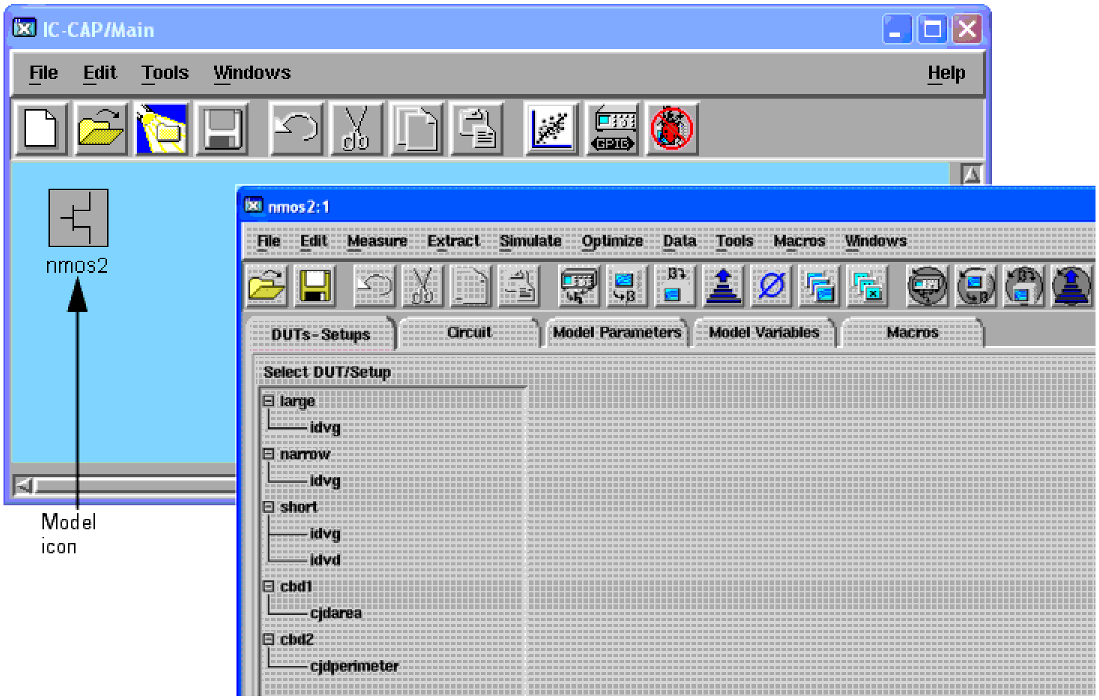
IC-CAP/Status window



To start the program and, at the same time, open a specific model file:

1. Change to the working directory.
2. At the system prompt, type `iccap` < *model filename* > and press Enter.
   Problem: When using the hpeesofsim simulator with the IC-CAP Optimizer, the message *Not a typewriter* is reported to the status window. The message is caused by a call to *stty -tostop* in the *hpeesofsim_start* script (*/bin/bootscript.sh*). This start-up script must be accessed before starting hpeesofsim. The message appears if *hpeesofsim_start* is customized by defining a different *HPEESOF_DIR*, or if it is used for a remote simulation. This message is benign, and can be ignored, or fixed if desired.
   Workaround: If the message is bothersome, it can be fixed by editing the file *HPEESOF_DIR/bin/bootscript.sh*. Change `stty -tostop` to
   ttymsg = "tty"
   if [! "$ttymsg" = "not a tty"]; then
   stty -tostop
   fi

The IC-CAP Main window and the IC-CAP Status window open. The IC-CAP Main window displays an icon of the model and the Model window opens.



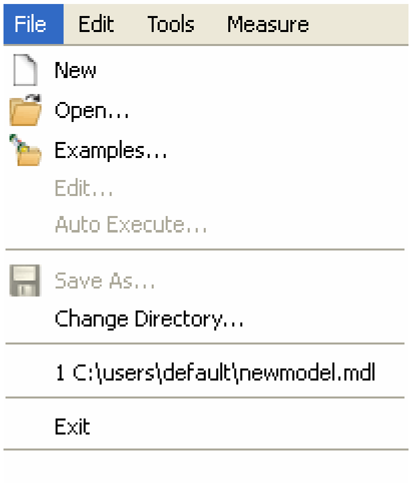
If the Model window does not open, you can open it by double-clicking on the Model icon.

> **Note**
> You can load a model and start a macro when you start the program. At the system prompt, type: iccap <*model_filename*> <*macro_name*>. For details, refer to *Autostart Macros* (prog).

Opening a Model File

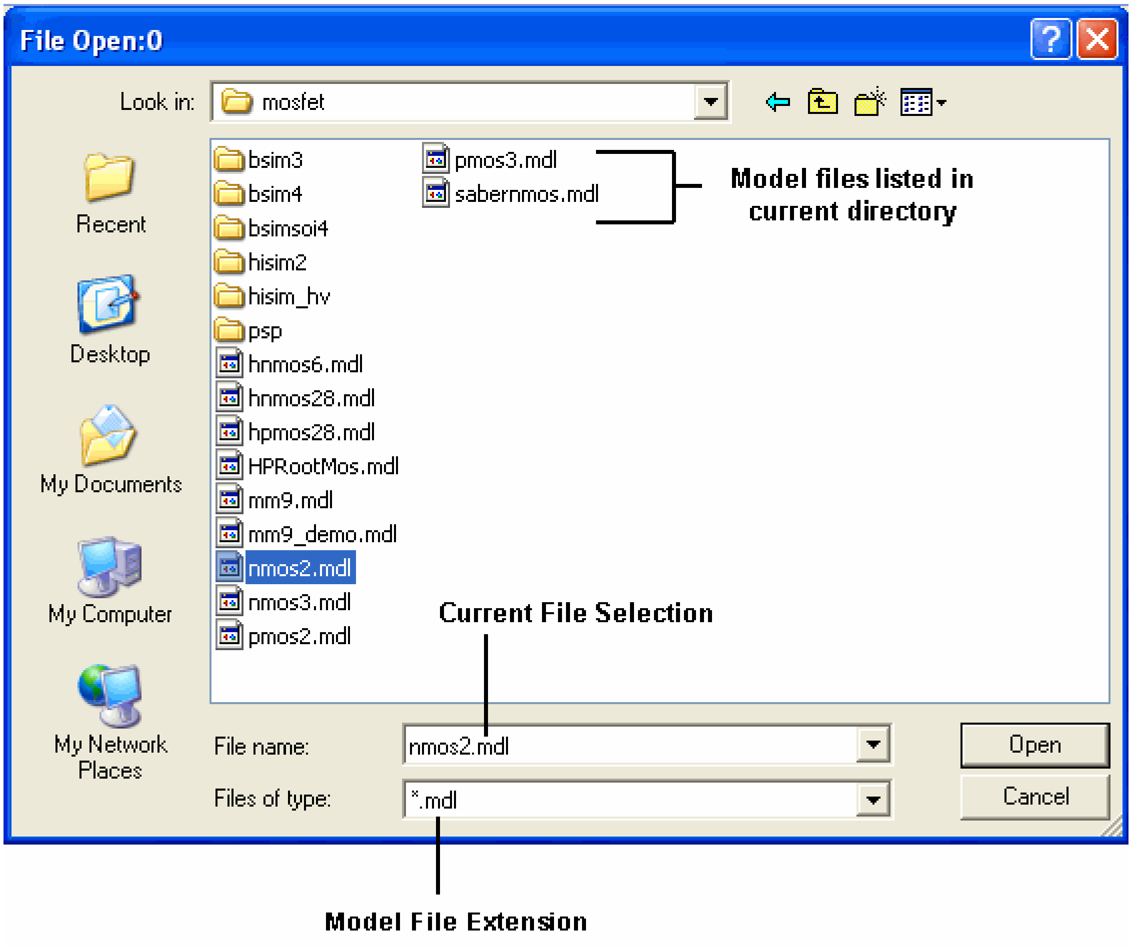
If you do not load a model file when you start the program, you can open the model file from the IC-CAP Main window.

To open a model file from the IC-CAP Main window:



- To open an empty model template, choose **File > New**.
- To open a model in the current directory, choose **File > Open**.
- To open a model provided with the program, choose **File > Examples**.

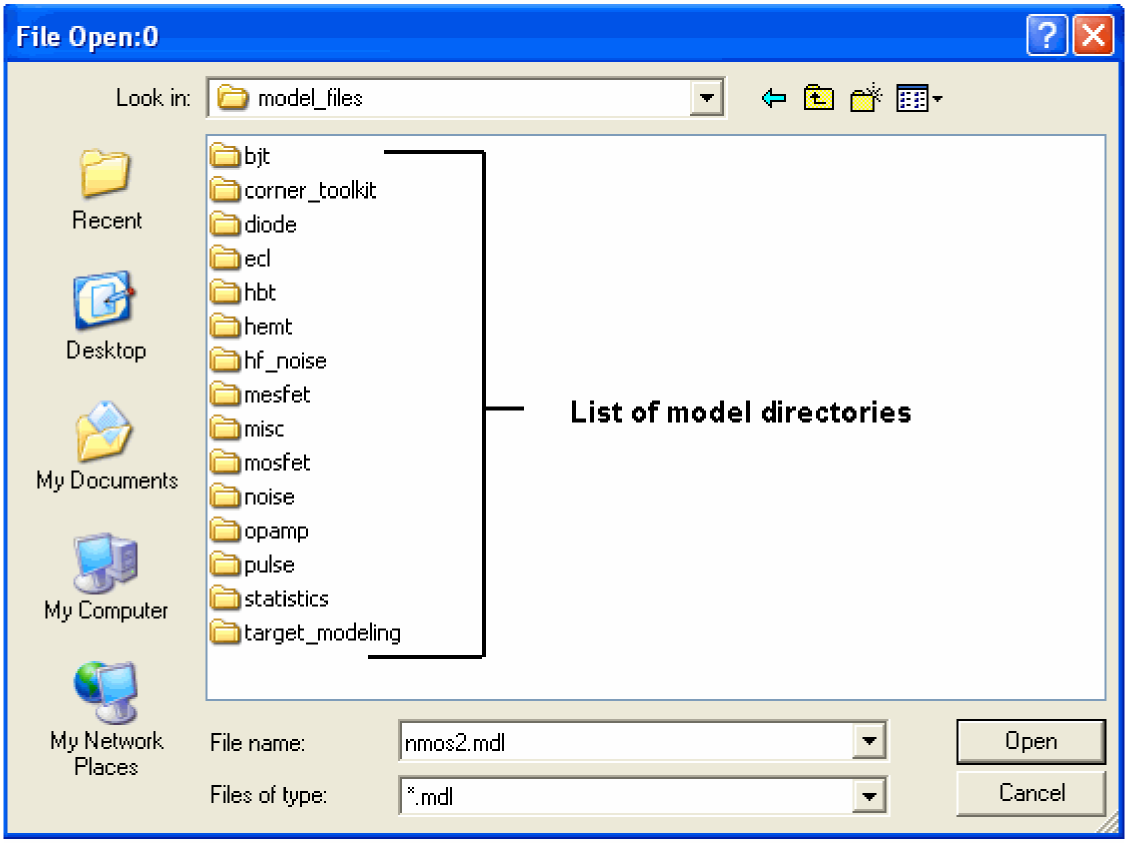
A directory dialog box is used when data is being written to or read from a file. Initially, the dialog box displays the files in the directory from which you started the session.



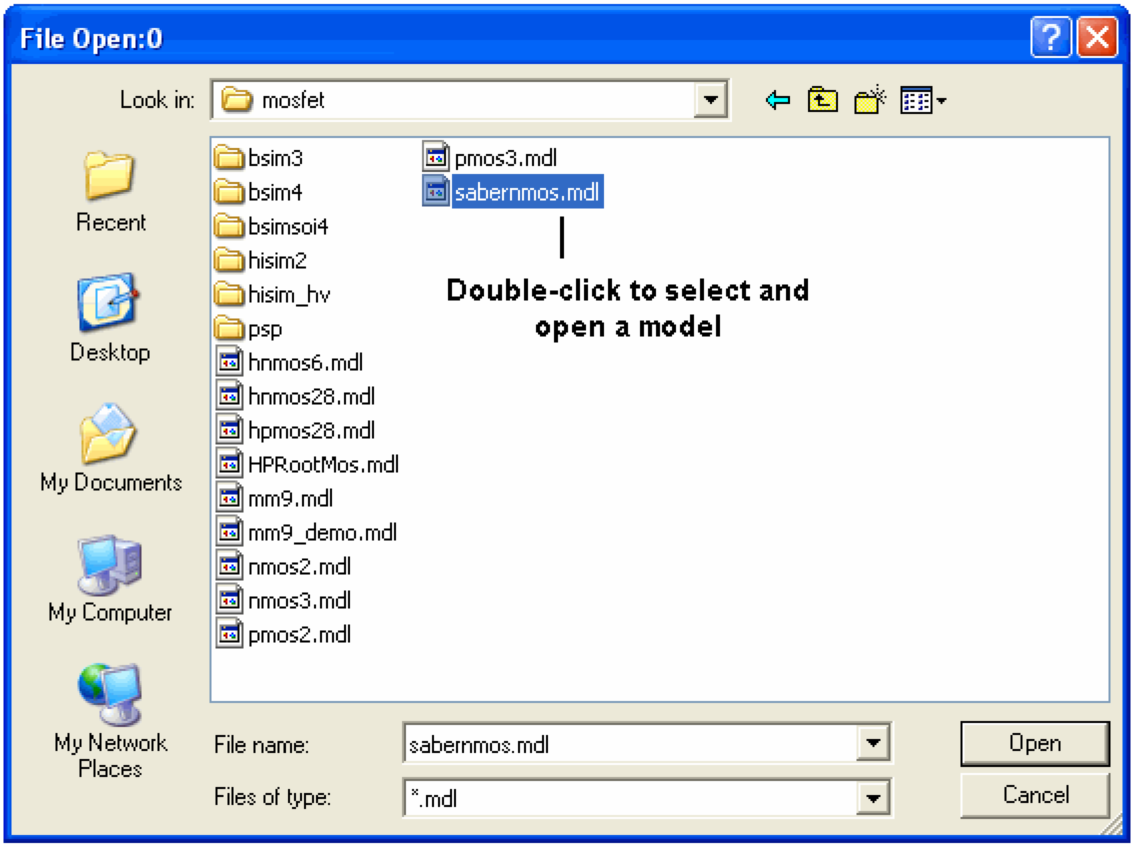
Note
On the PC, hidden and system folders do not appear in the Directory dialog box. If you can't see a directory, just change its attributes so it's not a hidden or a system file.

When you select *Examples*, double-click the **model_files** directory in the dialog box to display the directories of the models that are provided with the program (see following table). Double-click a directory to see a list of models. Double-click a model to select and open that model.

Example Directory Model Files



Example Model Files



$ICCAP_ROOT/examples/model_files/bjt:

HPEEbjt2.mdl	bjt_npn.mdl	hpsimnpn.mdl
bjt_ft.mdl	bjt_pnp.mdl	hpsimvbic.mdl
bjt_ncehf.mdl	hpsimbjt_nsehf.mdl	mnsnpn.mdl
bjt_nhf.mdl	hpsimbjt_nhf.mdl	mxt3t_npn.mdl
spectre_ncehf.mdl	spectrenpn.mdl	vbicsdd.inc
mxt4t_npn.mdl	mxt504_npn.mdl	sabernpn.mdl
vbic_npn.mdl		

$ICCAP_ROOT/examples/model_files/diode:

- HPDiode.mdl
- juncap.mdl
- pn_diode.mdl
- pn_test.mac

$ICCAP_ROOT/examples/model_files/ecl:

- ECLgate.mdl
- ECLornor.mdl

$ICCAP_ROOT/examples/model_files/hemt:

- HPEEhemt1.mdl

$ICCAP_ROOT/examples/model_files/mesfet:

- CGaas1.mdl
- CGaashfax.mdl
- UCBGaas.mdl
- CGaas2.mdl
- HPEEfet3.mdl
- UGaashf.mdl
- CGaashf.mdl
- HPRootFet.mdl
- hpsimHPEEfet3.mdl

$ICCAP_ROOT/examples/model_files/misc:

- 54120.demo.mdl
- lc.mdl
- plot_doc.mac
- TRLCAL.mdl
- npnwpnp.mdl
- sabercirc.mdl
- hp54750.mdl
- sys110_verify.mdl
- sys_test.mdl
- sys_testrf.mdl

$ICCAP_ROOT/examples/model_files/mosfet:

- mm9.mdl
- hpmos28.mdl
- mm9_demo.mdl
- HPRootMos.mdl
- hnmos28.mdl
- pmos3.mdl
- sabernmos.mdl
- hnmos6.mdl
- pmos2.mdl
- nmos2.mdl
- nmos3.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim3:

- BSIM3_DC_CV_Extract.mdl
- BSIM3_DC_CV_Measure.mdl
- BSIM3_RF_Extract.mdl
- Whats_New.mdl
- BSIM3_RF_Measure.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim3/tutorial:

- BSIM3_AC_Noise_Tutorial.mdl
- BSIM3_CV_Tutorial.mdl
- BSIM3_DC_Tutorial.mdl
- BSIM3_Temp_Tutorial.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim3/utilities:

- BSIM3_DC_CV_Finetune.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim4:

- BSIM4_DC_CV_Extract.mdl
- BSIM4_DC_CV_Measure.mdl
- BSIM4_RF_Extract.mdl|
- Whats_New.mdl
- BSIM4_RF_Measure.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim4/tutorial:

- BSIM4_DC_CV_Tutorial.mdl

$ICCAP_ROOT/examples/model_files/mosfet/bsim4/utilities:

- BSIM4_DC_CV_Finetune.mdl

$ICCAP_ROOT/examples/model_files/noise:

- noise_simu.mdl

$ICCAP_ROOT/examples/model_files/noise/1_f_toolkit:

- bjt_1f_noise.mdl
- mos_1f_noise.mdl

$ICCAP_ROOT/examples/model_files/opamp:

- mnsopamp.mdl
- opamp.mdl

$ICCAP_ROOT/examples/model_files/pulse:

- bjt_ncehfp.mdl
- opver_k46.mdl
- opver_k48a.mdl|
- cal_8510_p.mdl
- opver_k49.mdl
- opver_85124P.mdl|
- opver_k48b.mdl

$ICCAP_ROOT/examples/model_files/statistics/load_data:

bsim3_1.mdl	bsim3_4.mdl	bsim3_7.mdl
bsim3_10.mdl	bsim3_40.mdl	bsim3_70.mdl
bsim3_11.mdl	bsim3_41.mdl	bsim3_71.mdl
bsim3_12.mdl	bsim3_42.mdl	bsim3_72.mdl
bsim3_13.mdl	bsim3_43.mdl	bsim3_73.mdl
bsim3_14.mdl	bsim3_44.mdl	bsim3_74.mdl
bsim3_15.mdl	bsim3_45.mdl	bsim3_75.mdl
bsim3_16.mdl	bsim3_46.mdl	bsim3_76.mdl
bsim3_17.mdl	bsim3_47.mdl	bsim3_77.mdl
bsim3_18.mdl	bsim3_48.mdl	bsim3_78.mdl
bsim3_19.mdl	bsim3_49.mdl	bsim3_79.mdl
bsim3_2.mdl	bsim3_5.mdl	bsim3_8.mdl
bsim3_20.mdl	bsim3_50.mdl	bsim3_80.mdl
bsim3_21.mdl	bsim3_51.mdl	bsim3_81.mdl
bsim3_22.mdl	bsim3_52.mdl	bsim3_82.mdl
bsim3_23.mdl	bsim3_53.mdl	bsim3_83.mdl
bsim3_24.mdl	bsim3_54.mdl	bsim3_84.mdl
bsim3_25.mdl	bsim3_55.mdl	bsim3_85.mdl
bsim3_26.mdl	bsim3_56.mdl	bsim3_86.mdl
bsim3_27.mdl	bsim3_57.mdl	bsim3_87.mdl
bsim3_30.mdl	bsim3_58.mdl	bsim3_88.mdl
bsim3_31.mdl	bsim3_59.mdl	bsim3_89.mdl
bsim3_32.mdl	bsim3_6.mdl	bsim3_9.mdl
bsim3_33.mdl	bsim3_60.mdl	bsim3_90.mdl
bsim3_34.mdl	bsim3_61.mdl	bsim3_91.mdl
bsim3_35.mdl	bsim3_62.mdl	bsim3_92.mdl
bsim3_36.mdl	bsim3_63.mdl	bsim3_93.mdl
bsim3_37.mdl	bsim3_64.mdl	bsim3_94.mdl
bsim3_38.mdl	bsim3_65.mdl	bsim3_95.mdl
bsim3_39.mdl	bsim3_66.mdl	bsim3_96.mdl
bsim3_28.mdl	bsim3_67.mdl	bsim3_97.mdl
bsim3_29.mdl	bsim3_68.mdl	bsim3_98.mdl
bsim3_3.mdl	bsim3_69.mdl	load_stat_data.mdl

Note
Additional examples are located in *$ICCAP_ROOT/examples/demo_features* which contains examples that answer questions IC-CAP users ask most often. For information about the demos, open the *README.mdl* file in the */demo_features* directory.

IC-CAP File Extensions

Each file type has a unique file extension that is added to the filename automatically. The following table lists these file extensions.

IC-CAP File Extensions

Extension	File Type
.cir	Circuit
.dps	DUT Parameters
.ds	Dataset
.dut	DUT
.hdw	Hardware
.inp	Input
.iot	Instrument Options
.mac	Macro
.mdl	Model
.mdm	Data Manager
.mps	Model Parameters
.out	Output
.plt	Plot
.pop	Plot Optimizer
.set	Setup
.tci	Test Circuit
.vat	Variables Table
.xfm	Transform

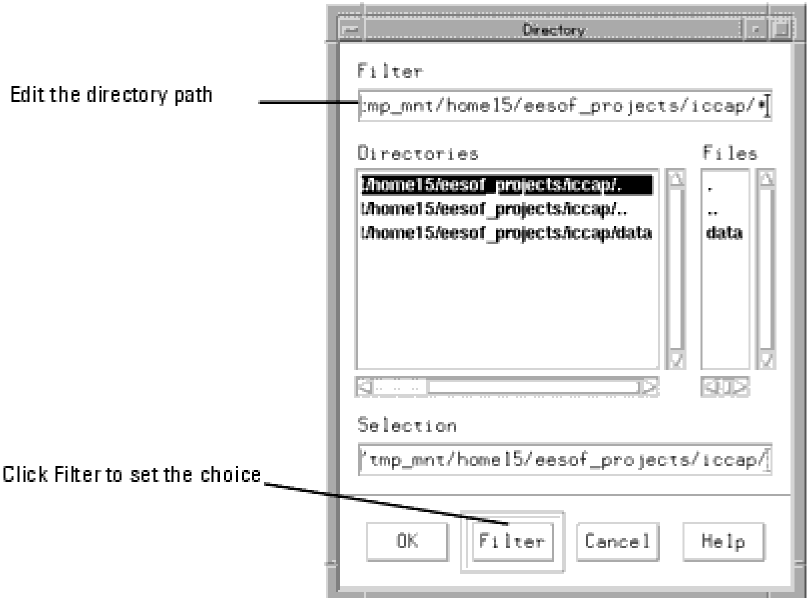
Changing Directories

When you load an example model, the directory path for the model icon in the IC-CAP Main window points to the installation directory. You can redirect the path to your working directory before saving the model data.

To change directory paths:

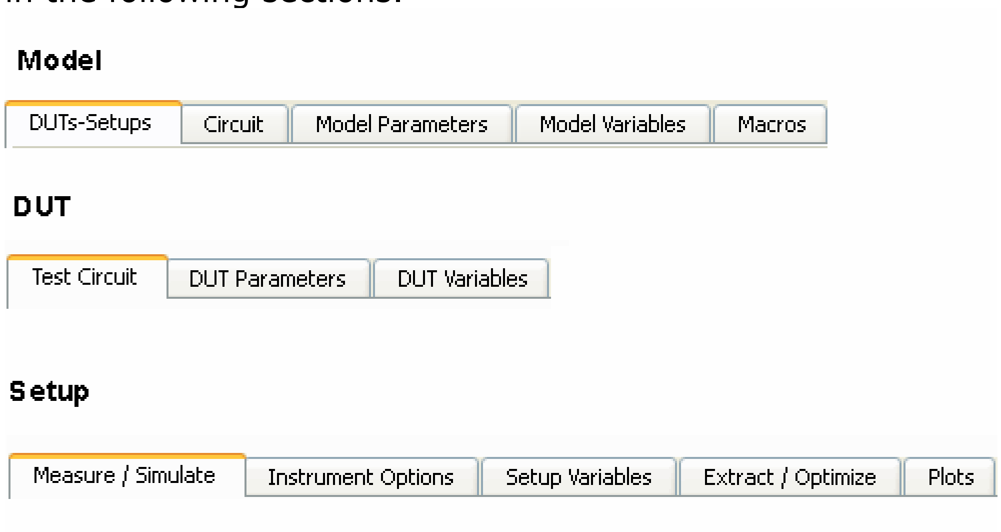
1. Select **File > Change Directories**.
2. Edit the selected path by typing a new name in the Selection field or by browsing the Directories list to the new directory.

   Note
   The path for IC-CAP model files cannot contain any folder names that use a space. For example, *C:\Model Files\IC-CAP 2004*. If a model file is saved in a folder name with spaces, you will not be able to open the model file. You will have to move the model file to a folder name that does not use a space.
   On the PC, hidden and system folders do not appear in the Directory dialog box. If you can't see a directory, just change its attributes so it's not a hidden or a system file.
3. Choose **OK** to set the selected path.



The Model Window Folders

The Model window contains sets of folders where you can view and edit the Model, DUTs, and Setups. This example describes each folder briefly. The folders are described in detail in the following sections.



Setting System Variables

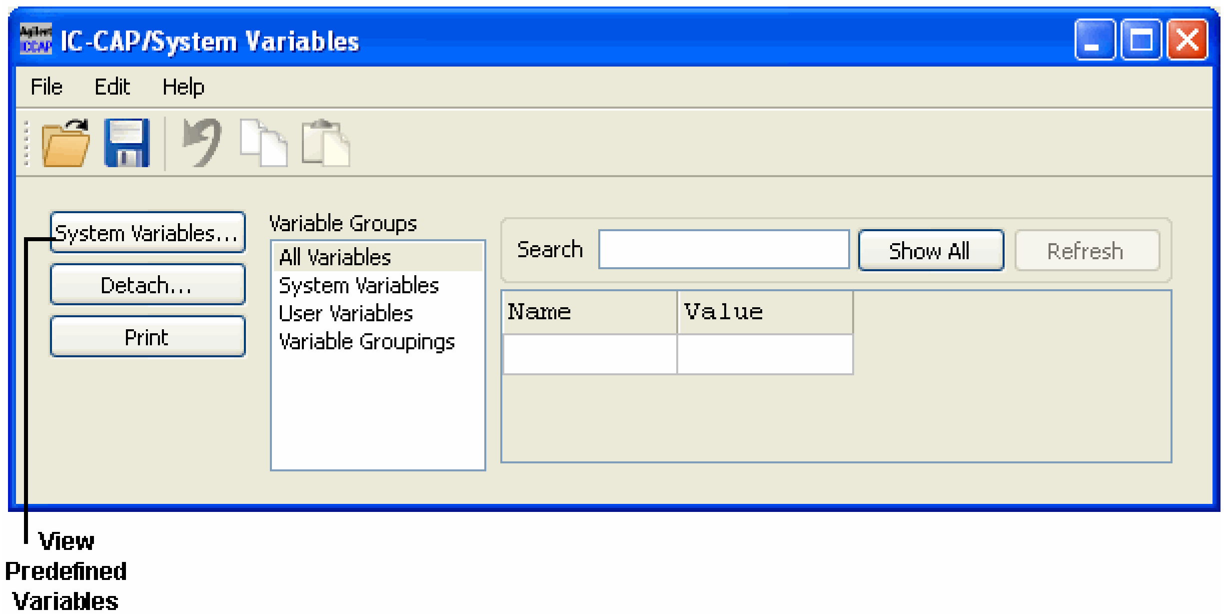
Variables can be defined at the top level (system) or at the Model, DUT, and Setup levels. You can define as many variables as you need and use these variables for other functions, such as extractions.

A set of predefined variables are supplied with the program. If you want to alter behavior of IC-CAP defaults and/or define new default behavior, you can set a system variable. Also, you can set a system variable to function as a *toolbox* for miscellaneous control parameters and save the set as a file so you can reuse it for different models.

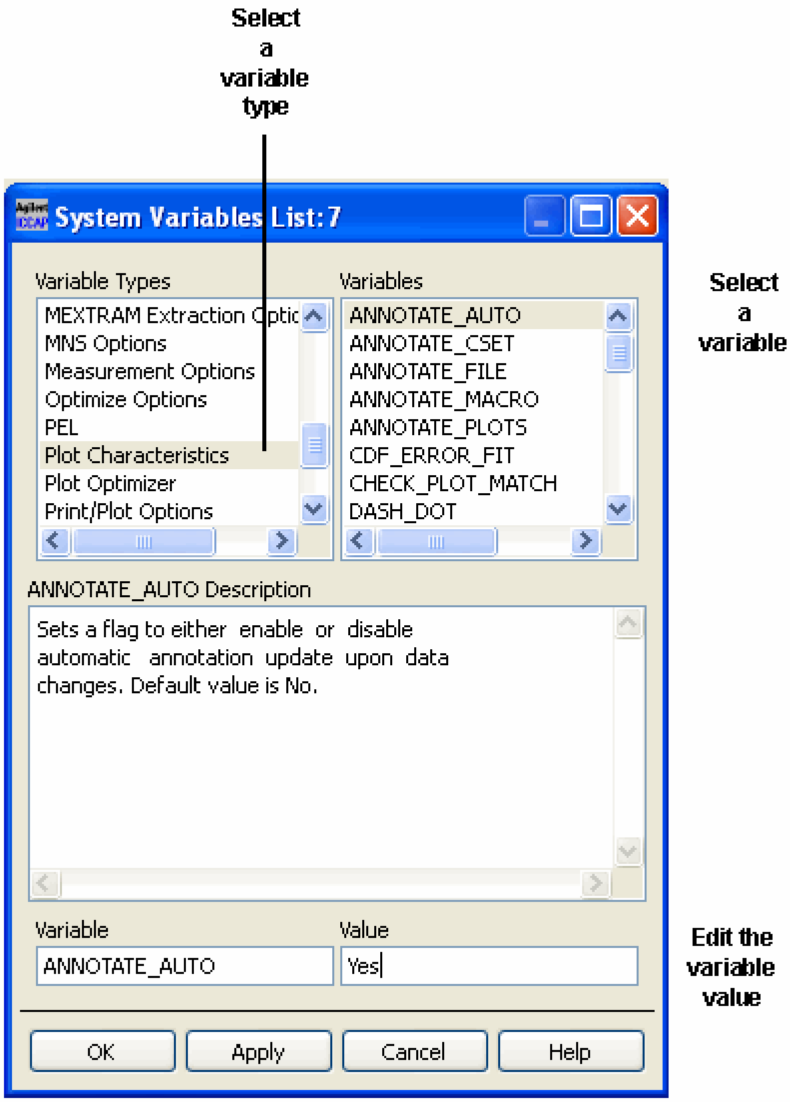
Note
For details on predefined system variables, refer to *Variables* (prog).

To view system variables:

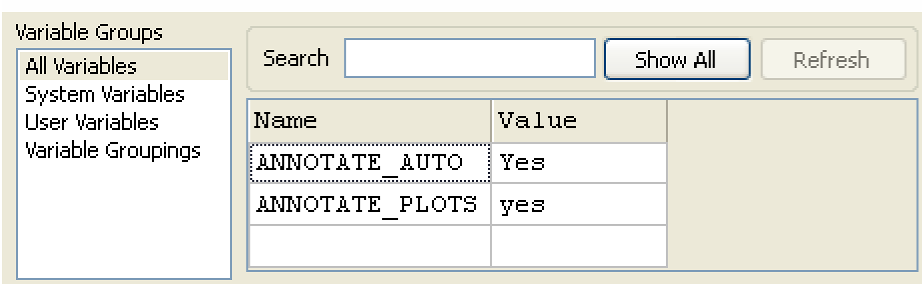
1. In the IC-CAP Main window, select **Tools**.
2. Select **System Variables**.
3. The System Variables window opens. Click **System Variables** to view predefined variables.



4. In the dialog box, select a Variable Type from the Variable Types list. Then select a Variable in the Variables list.



5. The selected variable and value appears in the Variable and Value fields. If appropriate, edit the Value field.
6. Click **Apply** to add the variable to the System Variables list.



7. When you have added all the system variables, click **OK** to close the dialog box.

Saving Model Data

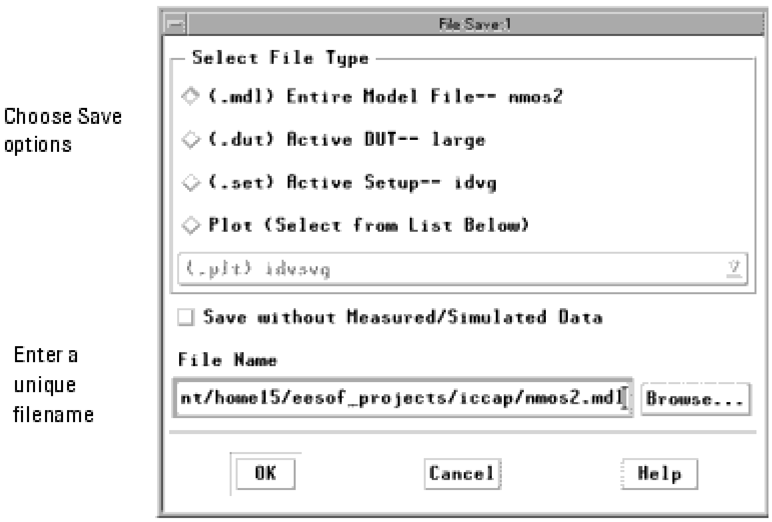
You can save IC-CAP information at several levels. For example, in the Model window:

- The parameter set can be saved before extractions are performed and restored if the extracted values are unacceptable.
- An entire setup can be saved for future restoration in order to make temporary changes to information such as input voltages.
- An output can be saved if you want to attempt another measurement but are not sure if the results will be improved.
- A snapshot of an entire model, or of all open models, can be made and used as a general-purpose backup.

> **Note**
> The path for IC-CAP model files cannot contain any folder names that use a space. For example, do not save to *C:\Model Files\IC-CAP 2006*. If you include a space in a folder name, you will not be able to load the model file. You will have to move the model file to a folder name that does not use a space.

To save model data from the Model window:

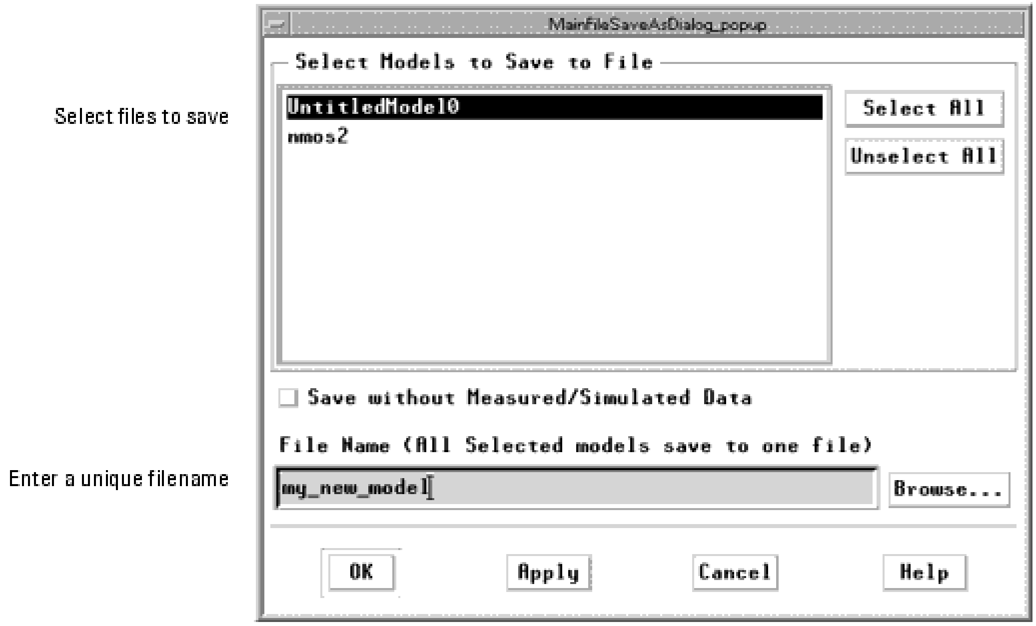
1. Select **Save**. 
2. Choose from the save options: save the entire model file, the active DUT, the active setup, or the active plot, and save without the measured/simulated data.



3. In the File Name field, type a unique filename and choose **OK**.

To save model data from the IC-CAP Main window:

1. Select **Save**. 
2. Select the open models you want to save.



3. In the File Name field, enter a unique filename and click **OK**.

> **Note**
> If a file was saved previously by that name, an information dialog box opens.
>
> To replace the existing model by that name, choose Yes.
>
> To avoid replacing the existing model by that name, choose No and enter a unique file name. Or use the *Save As* command to save the model to different name.
>
> You can use the same name as the supplied file, as long as you save the model to a different directory. For details on changing directory paths, refer to Changing Directories.

Tools

The Tools menu in the IC-CAP Main window provides access to a variety of features and options.

System Variables

Displays a window for viewing global (system) variables. For details, refer to Setting

System Variables and *Variables* (prog).

System GUI Items

Displays the System GUI Items window. For details, refer to *Creating Graphic User Interfaces* (prog).

Simulation Debugger

Displays the Simulation Debugger window. For details, refer to *Using the Simulation Debugger* (sim).

Stop Simulator

Stops any currently running piped simulator. The command is most useful when using the ADS simulator with the Root model to ensure correct simulations can be made after changing directories to test other models, and to allow model regeneration.

License Status

Displays a window with dynamically updated license information:

- The codewords currently in use
- The codewords currently available

To release a specific license, select it and click **Release**.

Hardware Setup

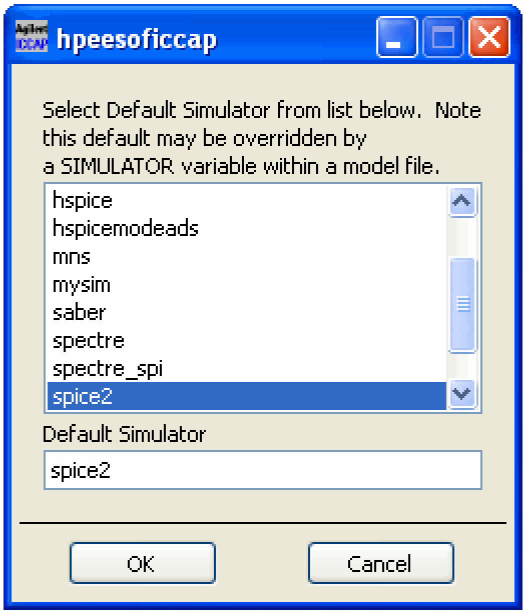
Displays the Hardware Setup window. For details, refer to *Configuring the System* (meas).

Statistics

Launches the IC-CAP Statistics program, if licensed. IC-CAP Statistics provides tools for identifying and analyzing the inter-relationships between device model parameters and electrical test data. For details, refer to *Statistical Analysis* (stat).

Select Simulator

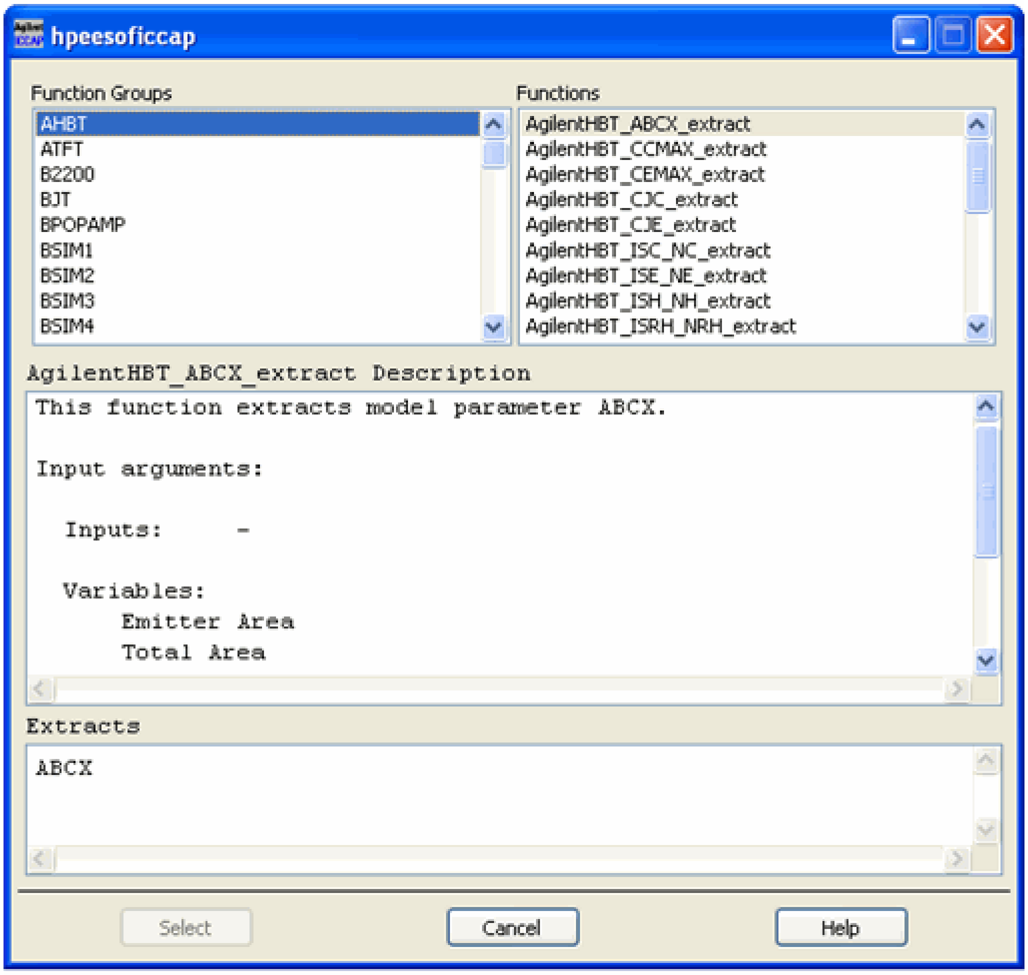
Displays a dialog for changing the default simulator after startup.



Note that this selection is overridden by a SIMULATOR variable. For details, refer to *Selecting a Simulator* (sim).

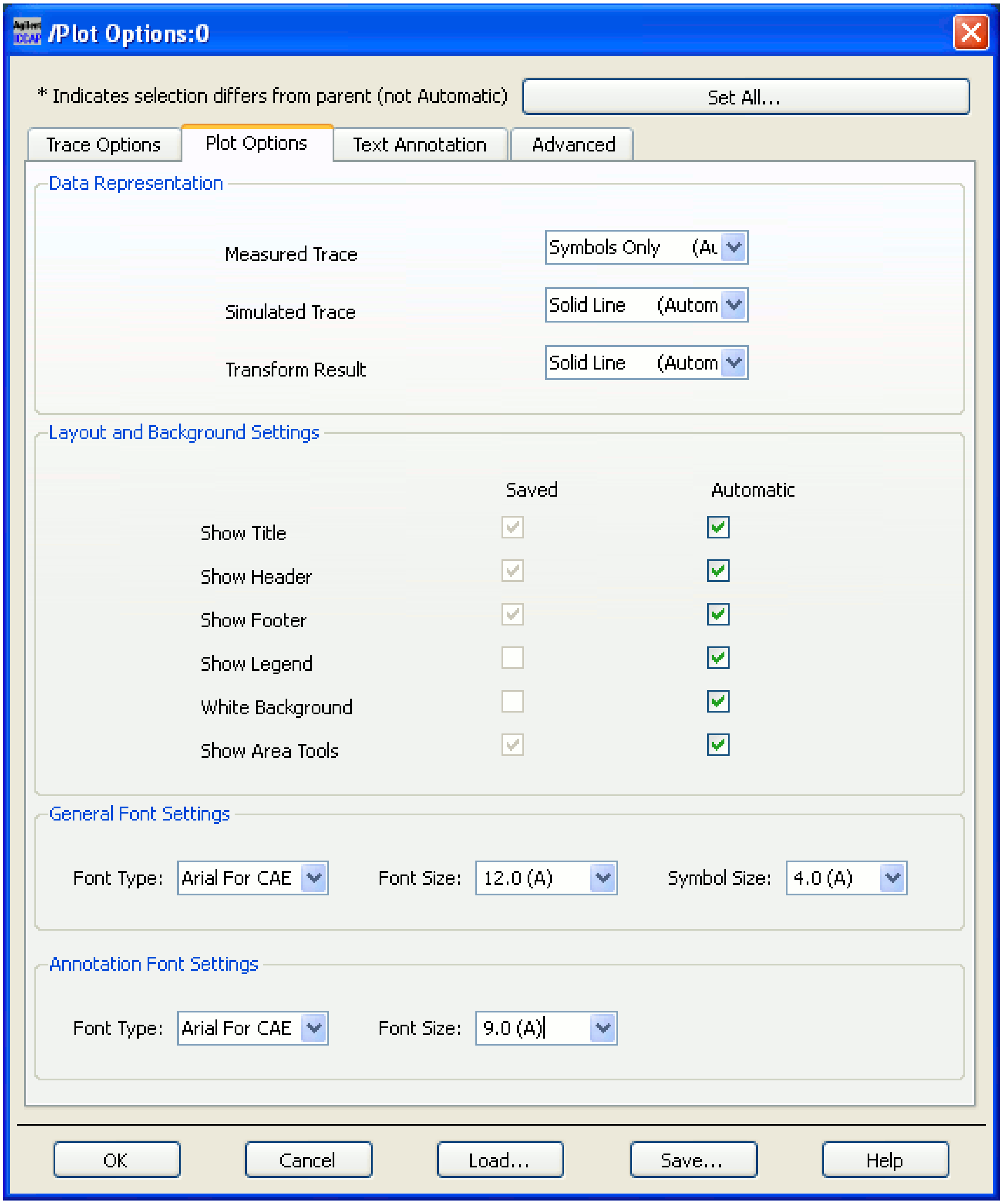
Functions

Displays the Function Browser dialog box for reviewing available functions. For details, refer to *Using Transforms and Functions* (prog).



Plot Options

Displays the Plot Options dialog box for defining trace options, plot options, and text annotation. For details, refer to *Setting Plot Options* (intro).



Options

Displays the following options:

- File Debug-Toggles the debugging facility on and off. When on, messages are recorded in the file .icdebug.
- Screen Debug-Toggles the debugging facility on and off. When on, messages are displayed in the IC-CAP Status window.
- View GUI Pages-Toggles displaying the GUI Items page on and off. When on, you will see a GUI Items page at the Model level, DUT level, and Setup level.
- Status Window to Top-If toggled on, the Status window pops to the front of the screen anytime new messages are displayed in it.
- Diagnostics-Executes internal diagnostics.

Aborting an Operation

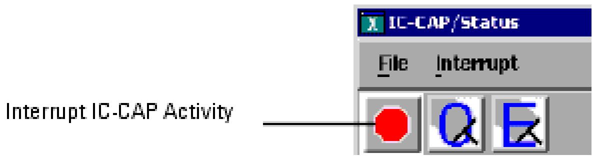
You can interrupt an IC-CAP activity from the Status window. The operations that can be aborted are: Measurement, Simulation, Macro Execution, Transform Execution, and Optimization. IC-CAP control is returned to you.

If you abort a measurement while an internal system sweep is in progress, the measurement in IC-CAP is aborted, but the instrument continues to step through its sweep values until the sweep is completed. If another IC-CAP measurement using this instrument is attempted before the sweep is completed, IC-CAP waits until the sweep is done before performing the measurement.

After an optimization is aborted, an optimization summary is printed before control is returned.

To abort an IC-CAP activity:

In the IC-CAP/Status window, click the Interrupt IC-CAP Activity button.



Alternatively, you can select the menu item, **Interrupt > IC-CAP Activity**.

Exit IC-CAP

To exit IC-CAP, select **File > Exit** in the IC-CAP Main window. This opens the *Save As* dialog box to give you an opportunity to save changes to models before exiting the program. Click on the *Exit* button to automatically close all open windows and terminate the program.

An Example IC-CAP Session

Using the IC-CAP Modeling System, you can characterize a device by following this general procedure:

1. Start the IC-CAP program and load the model file into memory.
2. Measure the device characteristics.
3. View the measured data.
4. Perform a simulation.
5. Extract the model parameters from the measured data.
6. Simulate with extracted parameters.
7. Compare measured data and simulated data.
8. Optimize model parameters as needed.
9. Measure the remaining devices.

Note
Your IC-CAP installation may not be identical to the system described. Depending on the options of your IC-CAP product, you may not be able to perform some portions of this demonstration. In such cases, an error message describes the missing codeword or license. For more information, refer to the installation procedures or consult your system administrator.

IC-CAP supplies various model extractions in model files which you can load and work with immediately. This section provides an example of a typical IC-CAP session, using a supplied model file.

Starting IC-CAP

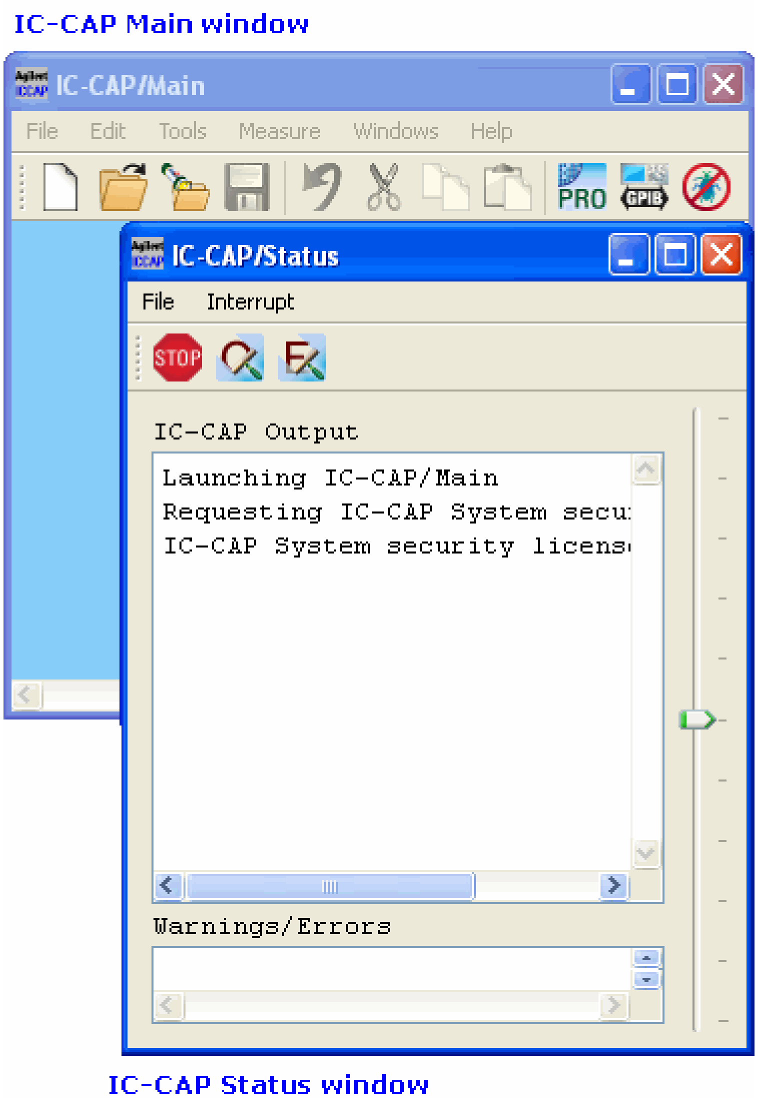
To start the program for the first time:

1. Change to the working directory.
2. At the system prompt, type `iccap` and press Enter.

   or

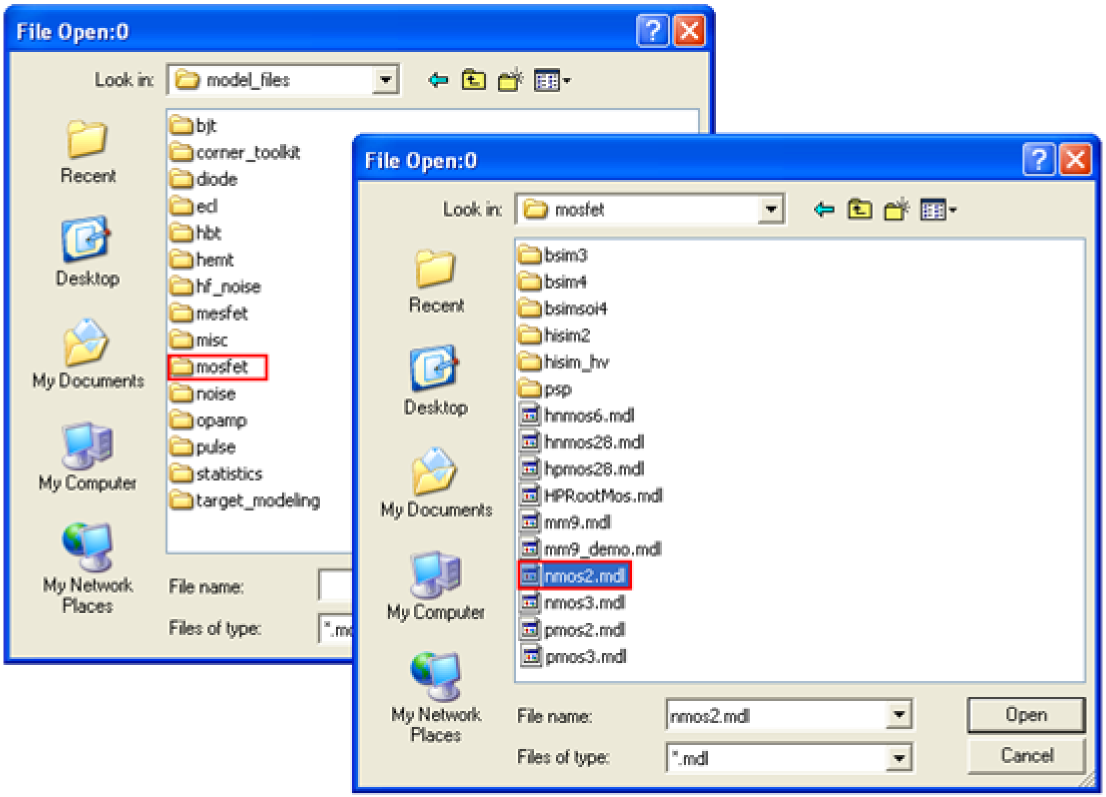
- Navigate to **Start > Programs > IC-CAP <version number>**.

The IC-CAP Main window and IC-CAP Status window open.

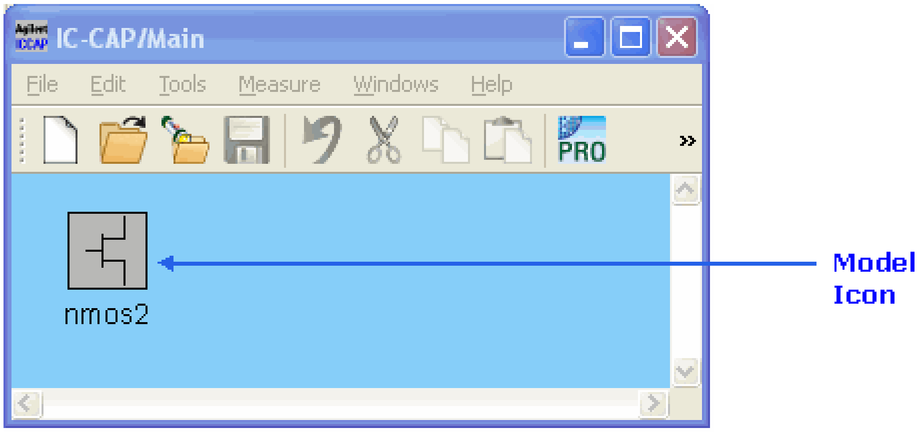


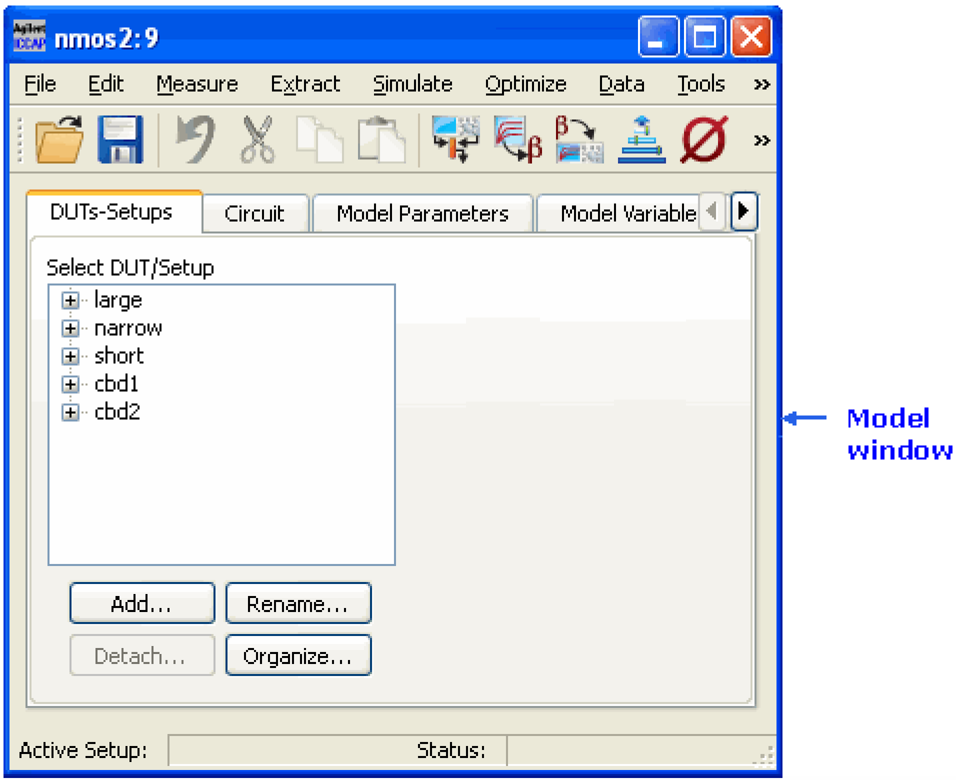
Opening a Model File

1. In the IC-CAP Main window, choose **File > Examples** or click **Examples** icon  on the toolbar. The File Open dialog box displays the directories of model file examples.
   The *model_files* directory is located in your IC-CAP installation directory under the `IC-CAP installation path\ICCAP_2010_08\examples\model_files`.
2. Double-click *model_files* directory to view the categories of the models that are provided with the program.



3. Double-click the *mosfet* directory. Select **nmos2.mdl** and click **Open** to open the example nmos2.mdl model file. An icon of the model file is displayed in the IC-CAP Main window and the Model window opens.
   The **nmos2.mdl** example extracts the parameters for Level 2 N-channel UCB MOSFET transistor.



3. 

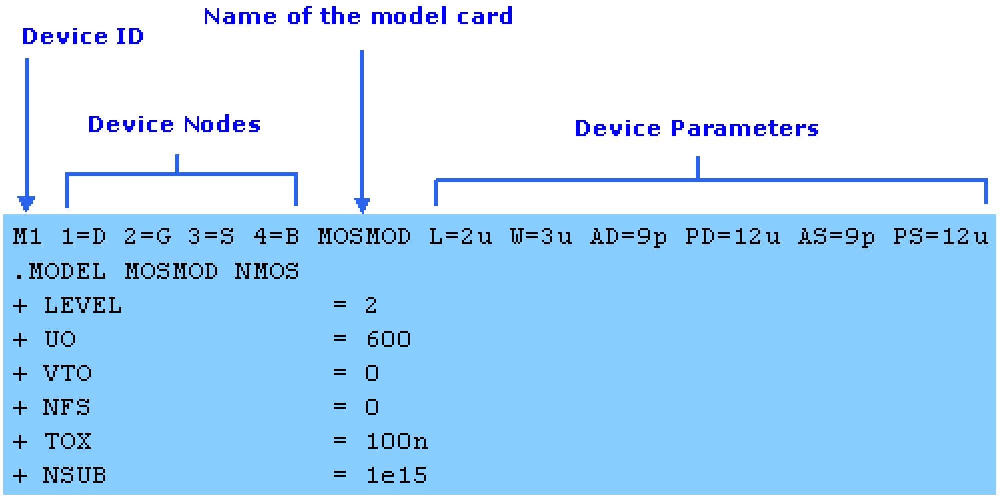
Viewing the Circuit Description

The **Circuit** description defines the device topology and parameters to be extracted.

To view the default values of the parameters and circuit definition, click **Circuit** tab in Model window.

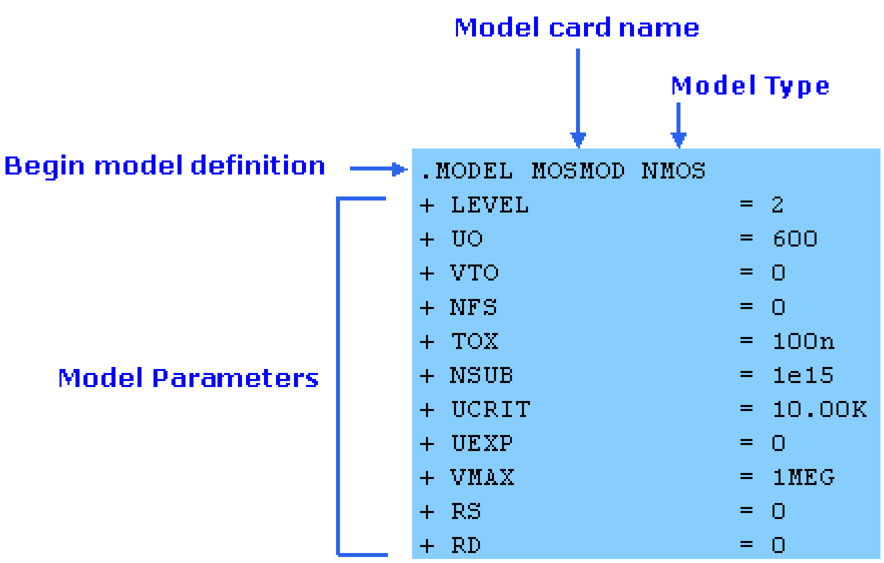
The circuit definition begins with the Options definition `(.OPTION)`. The definition lists the device to be measured, the device nodes, the name of the model card that contains the model parameters used by the device, and the device parameters.

Circuit Options Definition



The next section is the model card definition (.MODEL). The definition lists the name of the model card, the model type, and the contents of the model card. The contents of the model card consists of the parameters that are to be extracted (model parameters) and their assigned initial values.

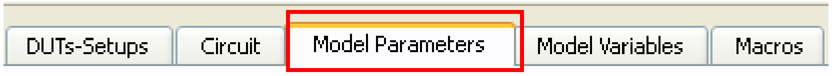
Circuit Model Definition



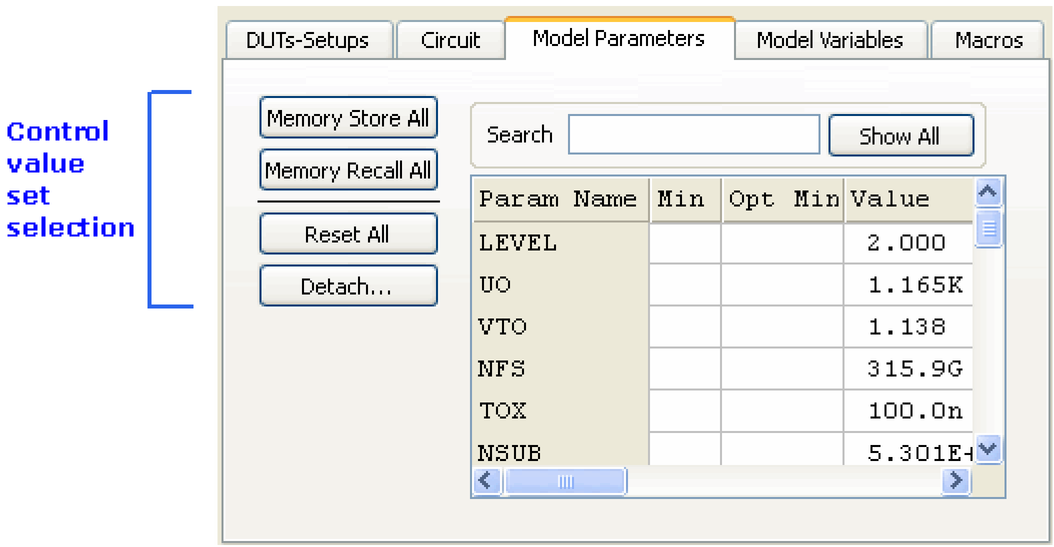
Defining Model Parameters

The **Model Parameters** displays the current parameter values. These parameters are common to all DUTs for that model and are used in a simulation. When a new model is read, these values are set to the defaults defined in the circuit. The values change after extractions and optimization are performed.

To view and define the model parameters, click **Model Parameters** tab in the Model window.



The Parameters table contains the same names as those specified in the *MODEL* definition of the Circuit. However, by editing the table, the values can be set independently of those in the circuit definition and can be specified as a real number or an equation. In addition, you can set limits by entering minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values.



After editing the defaults, click **Memory Store All** to store the current set of parameter values in a temporary buffer or **Memory Recall All** option to recall these values anytime during the IC-CAP session. If you prefer, you can also restore the default values by clicking **Reset All**.

Note
The values stored using **Memory Store All** option are not saved in the model file and are lost when the IC-CAP session ends.

Defining Model Variables

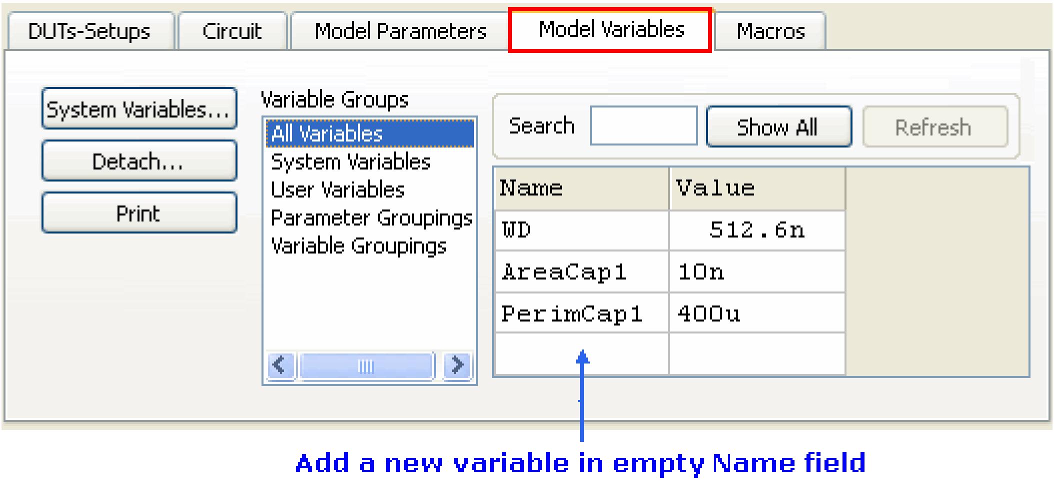
Model Variables can be defined at the top, system, or at the Model, DUT and Setup levels. You can define as many variables as required and use these variables for functions, such as extractions. Model variables can be accessed by any of the components of a model, including the DUTs and setups, while DUT and Setup variables (such as inputs or outputs) can be accessed only by the components of that setup. The Model Variables contains the names and values of all variables that are global to the model.

A useful example of a variable defined at the model level is the compliance limit for measurements. Typically, compliance values are defined for each input in a setup.
You can refer all of the compliance values in a model to one variable defined at the model

level by creating a compliance variable for the model. This compliance variable can be modified to change the compliance values for all inputs in the model.

To view the current values of the model variables, click **Model Variables** tab in the Model window. To view system defaults from the Model Variables folder, click the 

button.



Defining a New Variable

To define a new variable:

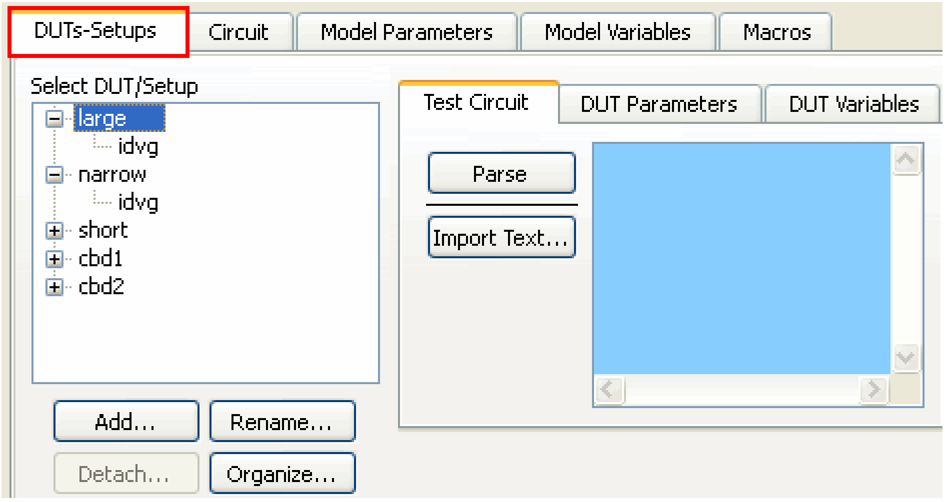
1. Type the variable name in the first empty **Name** field row.
2. Press **Tab** to move the cursor to the **Value** field. Type a value for the variable.
3. Press **Enter** to accept the value and open a new row.

> Do not attempt to enter an expression when editing model variables. Expressions in variables are not evaluated when the variables are referenced. If a variable is used in a numeric expression, enter a single number only.
> Do not enter the name of a variable table variable into the value field. The value of the variable is not evaluated when the variable is referenced.

Defining DUT Options

IC-CAP models usually contain several DUTs. DUTs contain groups of setups that have a similar physical connection to the device. Each DUT contains its own DUT parameter set and test circuit. If two setups require differences in either of these areas, you must define a different DUT for each. Before you can measure a device, you must make any needed changes to the DUT options. The **DUT-Setup** tab displays the DUTs and setups in the model. You can define DUT options through **Test Circuit**, **DUT Parameters**, and **DUT Variables**.

To view the DUT options and make a device active, click **DUTs-Setups** tab in Model window. Then, select the DUT (for example, large). You can view the setup list by expanding the DUT.



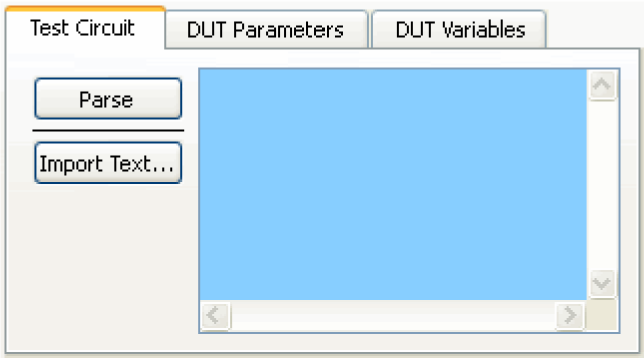
Accelerator Pop-Up Menus

In the **DUT/Setup** tree structure, you can view pop-up menus which allow you to perform actions on DUTs and Setups easily. To view the menus, right-click with your mouse pointer over a DUT or Setup name. The menus for a DUT and a Setup are different. The commands available in the menus enable you to perform selected actions that are allowed for the DUT/Setup. These are the same actions that are available in the Model window's main menu bar and folders. The pop-up menus save time by enabling you to perform an action on a DUT or Setup while avoiding the time required to display the folder contents in the work area.

As you work through this example, try also using these pop-up menus to perform the actions. The contents displayed in the Model window's work area depends on the DUT/Setup folder currently displayed in the work area, and which pop-up menu item you right-click on. When you right-click on a DUT or Setup, it is selected just as if you left-clicked on it. For a DUT or Setup folder displayed in the work area, if you perform an action from the pop-up menu normally associated with a different folder, the associated folder is not displayed in the work area. If a folder is currently shown, and you right-click on a different DUT or Setup, the current folder closes so there is no inconsistency between the selected DUT or Setup and the work area.

Defining a DUT Test Circuit

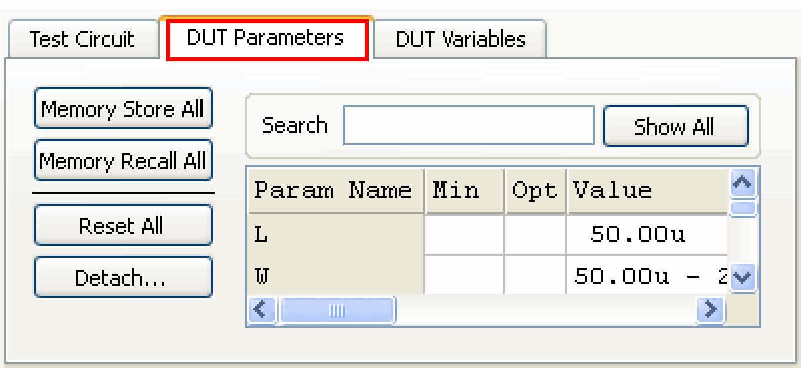
You can use the **Test Circuit** tab to attach additional circuitry between the device or circuit being modeled and the measurement hardware. For example, an OpAmp may need to be connected in different configurations for different measurements. For complete information on defining Test Circuits, refer to *Simulating* (sim).



Defining DUT Parameters

You can edit the values of parameters that change from one DUT to another (such as, channel length and width for a MOSFET). In addition, you can set limits by defining minimum and maximum values for each parameter. If a parameter value is outside its limits, it will be clamped to its minimum or maximum value. After editing the values, you can temporarily store the current set of parameter values (**Memory Store All**), recall stored parameter values (**Memory Recall All**), apply the changes to the circuit definition (**Update Circuit**), reset the default values specified in the circuit definition (**Reset All**).

To view the DUT parameters, click **DUT Parameters.** The **Import Text** option allows you to reuse a circuit text file.

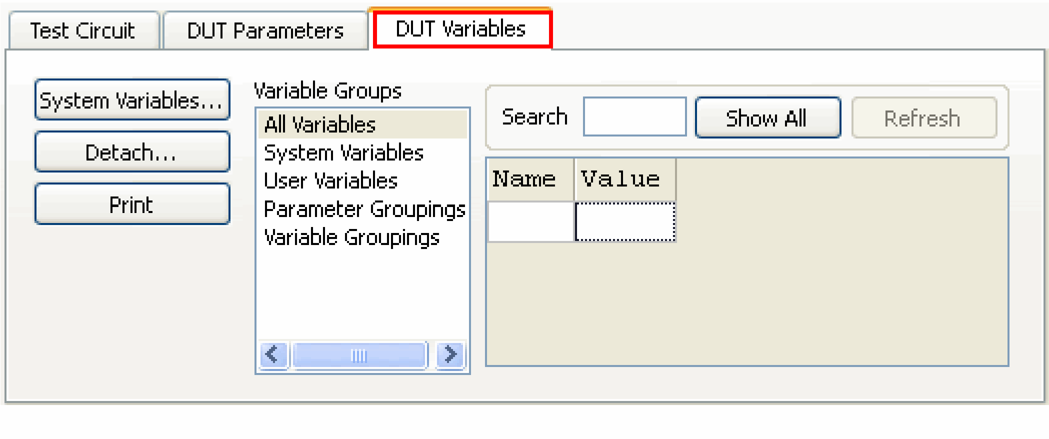


Defining DUT Variables

The **DUT Variables** tab contains the names and values of all variables that are global to the DUT. To view system defaults from the DUT Variables, click the 

button.

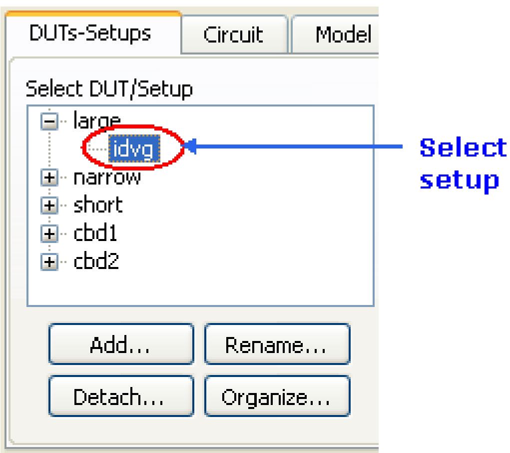
To view the DUT variables, click **DUT Variables**.



Defining Setup Options

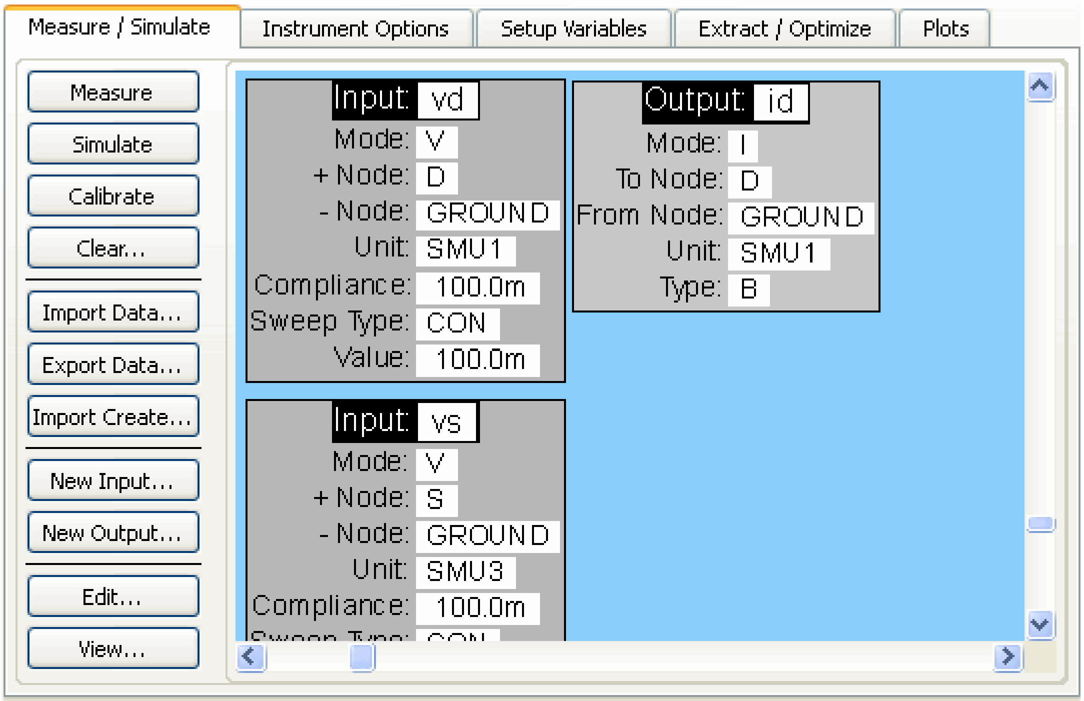
You can define setup options from the DUT-Setup panel. A setup contains definitions for inputs, outputs, instrument options, measuring and simulating, and results.

To view and edit a setup, select a setup from the DUT-Setup panel.



To detach the setup window for convenient editing, click **Detach** below the DUT/Setup tree.

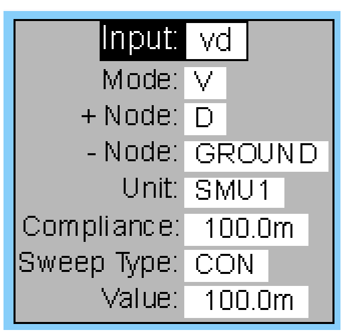
The Measure/Simulate window allows you to define the inputs and outputs for a particular setup.



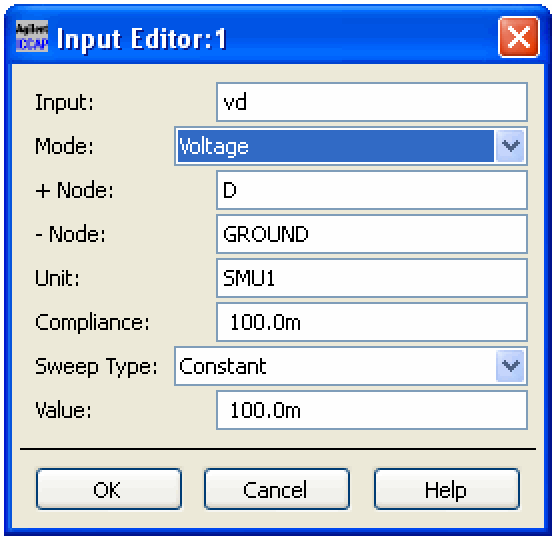
Viewing or Editing Input or Output

Setup input and output data are listed in a table (sometimes called a *tile*). You can edit input and output in one of the following two methods:

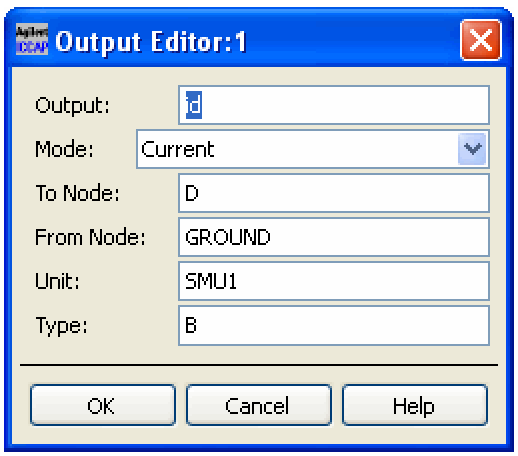
- **Method 1** - Edit directly in the table. Click or double-click a field and type the new data.



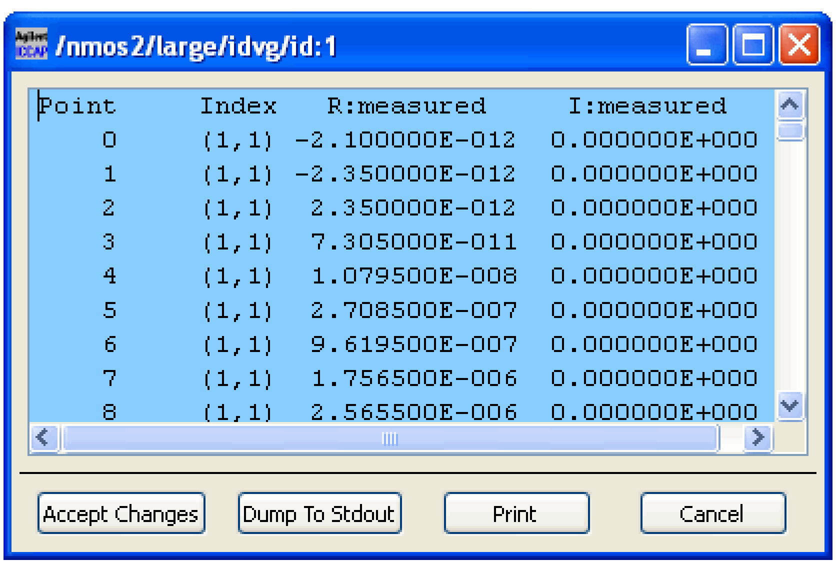
- **Method 2** - Use the Input or Output Editor. To open the Input or Output Editor, first select the table and then click **Edit**. Edit the fields and click **OK.**

- 

Output parameters can be derived from Measured data, Simulated data, or Both.



To view the raw data for an input or output, select one or more tables then click **View**.

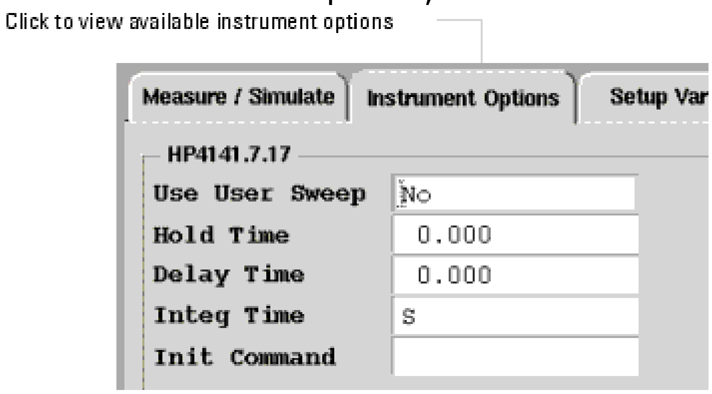


> **Note**
> You can enable the Input/Output Finder mode in the Measure/Simulate folder. The Input/Output Finder mode adds an area that lists the Setup's inputs and outputs. When enabled, you can quickly bring data into view by selecting one or more of the inputs and outputs in the list.
>
> To enable the mode for a Setup, open the Setup Variables folder. Enter the variable name SHOW_INPUT_OUTPUT_FINDER in the Name field, then enter yes, true, or a non-zero integer in the Value field. If you change the variable's setting after opening the Model, you must close (File > Close), then reopen, the Model window for the new setting to take effect.

Viewing or Editing Instrument Options

The Instrument Options table is available after the hardware has been set up.

To edit instrument options, select the **Instrument Options** folder.

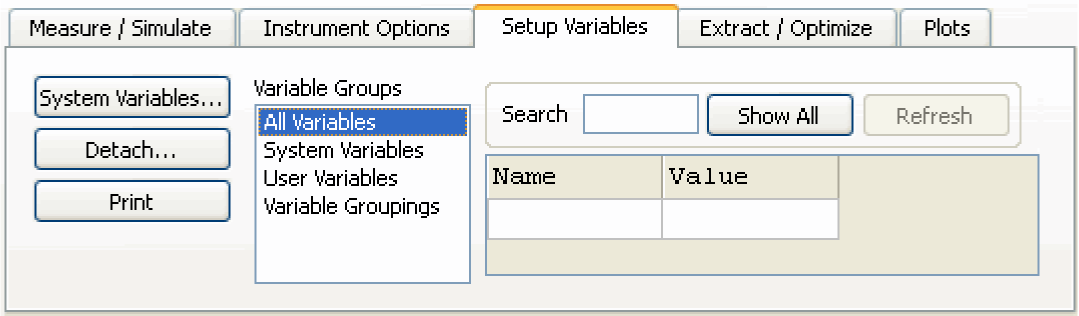


For details on Hardware Setup, refer to *Configuring Hardware and Performing Measurements* (meas).

Defining Setup Variables

The **Setup Variables** tab contains the names and values of all the variables that are global to a Setup.

To view the Setup variables, click **Setup Variables** tab in the Model window.



Measuring the Device

The measurement on the devices provides sufficient information for extracting a full set of model parameters. To avoid errors, measuring devices, and extracting parameters must be performed in a specified order.

> **Note**
> For detailed procedures on measurement order, refer to the tutorial for your model.

- To initiate the measurement using the active setup, select the setup and click **Measure Setup** icon  .
- To initiate the measurement using all setups in the active DUT, select the DUT and choose **Active DUT** from **Measure** menu in the Model window.

Performing a Simulation

You can determine the accuracy of current parameter values by performing a simulation and comparing the results to the measured data.

- To perform a simulation on the active setup, select the setup and click **Simulate Setup** icon  .
- To perform a simulation on all setups in the active DUT, select the DUT and choose **Active DUT** from **Simulate** menu in the Model window.

Extracting Model Parameters

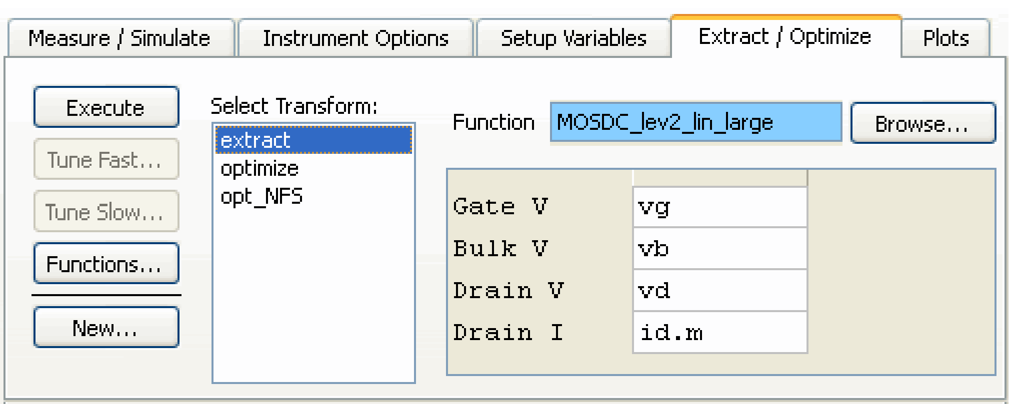
Agreement between two sets of data is not always good prior to extraction. IC-CAP modules provide standard device extractions, as well as macros for performing an extraction. If the model you are using does not have macros available, you can set the criteria in the Transforms folder.

Viewing or Editing Transforms

You can choose from a large variety of functions to use for extracting model parameters in a Transform.

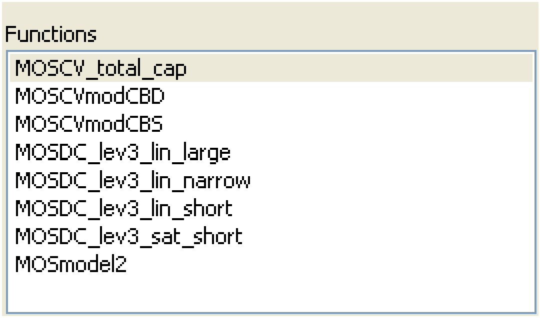
To view or edit transforms:

1. Select a setup.
2. Choose the Transform as **extract** under the **Extract/Optimize**.



3. Click **Browse** to view the Function list.

4. Select a Function from the **Functions** list and click **Select**.



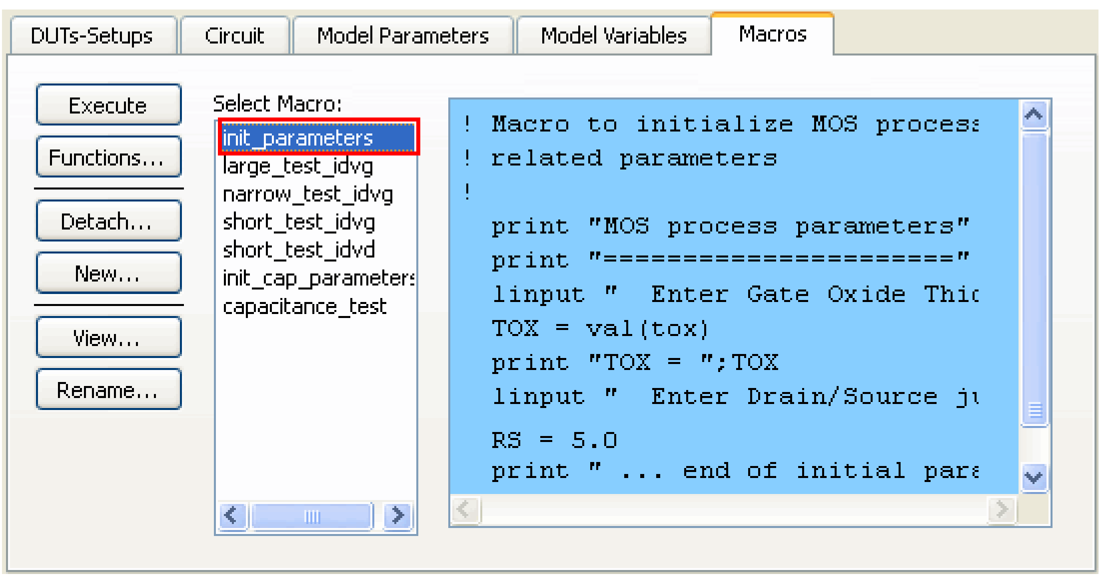
Setting Initial Values Using a Macro

The initial values of the process-dependent parameters for the extraction are entered using a macro.

To run a macro:

1. Click **Macros** tab in Model window.

2. Choose **init_parameters** from the *Select Macro* list and click **Execute**.



> You can also execute the macros from the Model window's main menu. Click **Macros > Execute** and then select the macro you want to run.

3. Enter the initial values in the series of prompts.

3. 

Performing an Extraction

You can extract all extraction transforms in the active setup or extract all setups in the active DUT.

- To perform an extraction on all extraction transforms in the active setup, select the setup and click **Extract Setup**  .
- To perform an extraction on all extraction transforms in all setups in the active DUT, select the DUT and choose **Active DUT** from **Extract** menu in the Model window.

When you extract all the setups in an active DUT, the extractions are performed in the left-to-right order listed in the setup. This order is usually critical to proper extraction performance.

> **Note**
> For detailed procedure on extraction order, refer to the tutorial for your model.

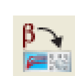
Typically, extractions are completed instantly. The newly extracted model parameter values are listed in the IC-CAP Status window and are placed in Model Parameters.

```
MOSDC_lev2_lin_large Extraction Results:
        VTO =  -862.7m
        NSUB =   38.49T
        UO =   1.056MEG
        UEXP =   2.289
        VMAX =   1.000MEG
        NFS =   1.000T
```

Simulating with Extracted Parameters

After extracting the model parameters, you can perform a simulation with the extracted parameters and compare the results to the measured data.

To perform a simulation:

- On an active setup, select the setup and click **Simulate Setup** icon  .
- On all the setups in the active DUT, select **Active DUT** from **Simulate** menu in the Model window.

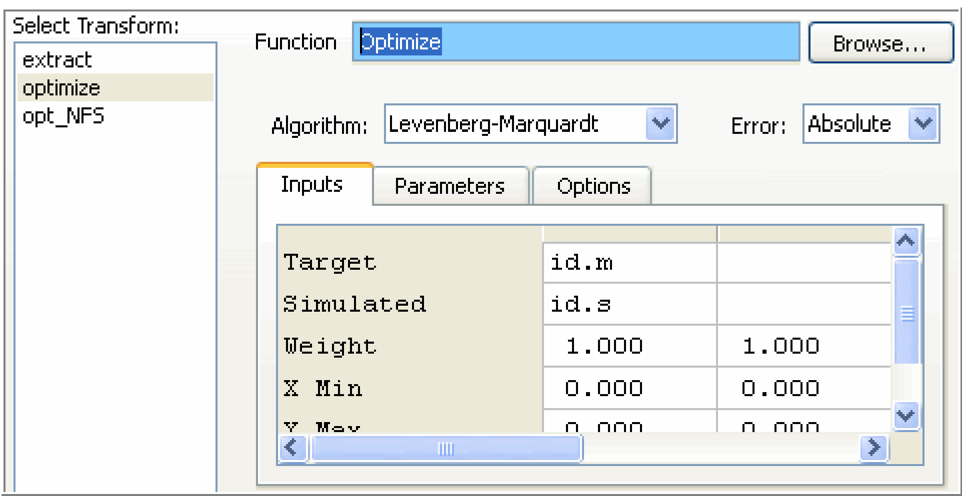
Repeat the simulation procedure for each DUT in a model. You can observe the differences in output that result from changes to various model parameters. For example, change the value of one or more parameters. Then, run a simulation and view the changes in the plot.

Optimizing the Model Parameters

To achieve greater accuracy, you can optimize the model parameters. This optional step obtains the best possible fit between measured and simulated data by altering the parameter values iteratively until the difference between the data sets is minimized. Since repeated simulations are required, optimization is more time-consuming than an extraction.

To perform an optimization:

1. Select a setup (for example, idvg).
2. Select the Transform as **optimize** under the **Extract/Optimize** tab.
3. Define the **Inputs**, **Parameters**, and **Options** for the selected Transform. For more details on these Transform parameters, refer to *Using Transforms and Functions* (prog).



4. To perform an optimization on an active setup, select the setup and click **Optimize Setup** icon  .
5. To perform an optimization on all setups in an active DUT, select **Active DUT** from **Optimize** menu in the Model window.

Measuring the Remaining Devices

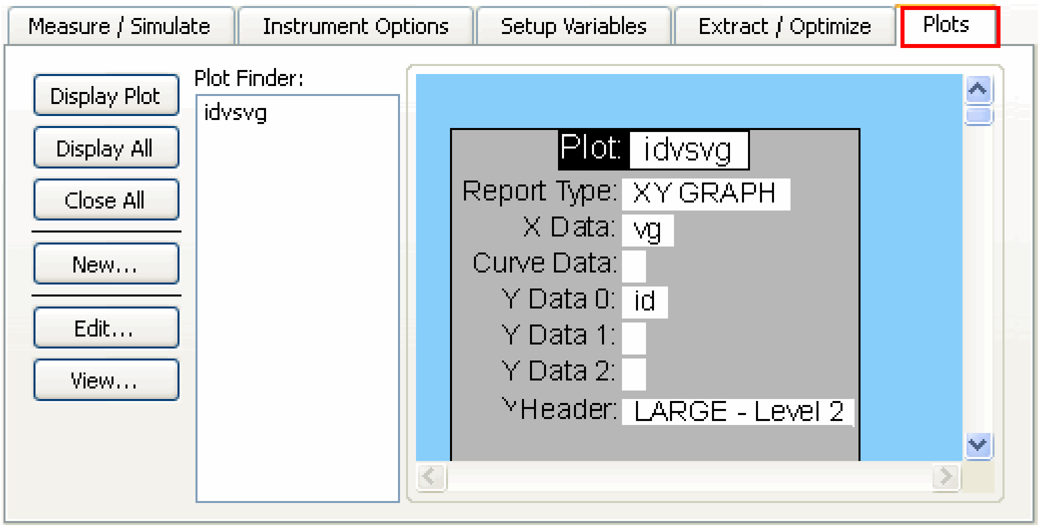
To complete the characterization, you must measure the remaining devices by repeating these steps for the other DUTs present in the model.

To measure:

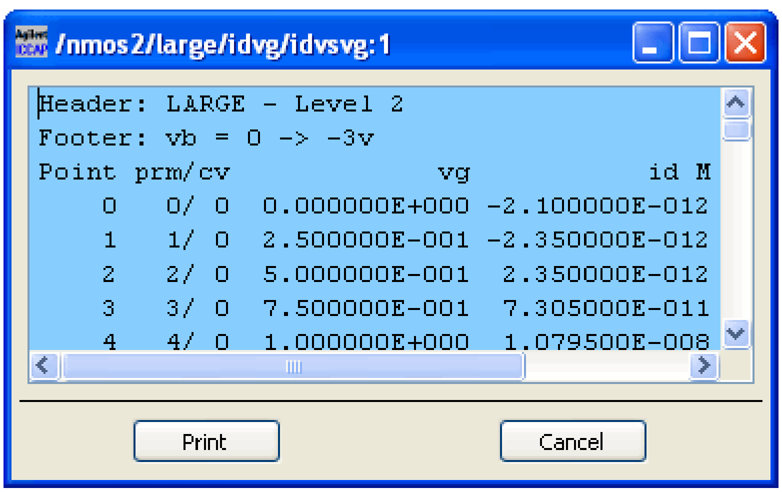
- All the setups in a DUT with multiple setups, select **Active DUT** from **Measure** menu in the Model window.
- A setup in a DUT, click **Measure DUT** icon  on the Model window toolbar.

Viewing the Results

Click **Plots** tab in the Setup window to view the results of the measured data. The **Plot Finder** lists the plots available for a Setup. If there are multiple plots, select a plot you want to view. You can view multiple plots by pressing the **Ctrl** key while selecting the plot names in the list.



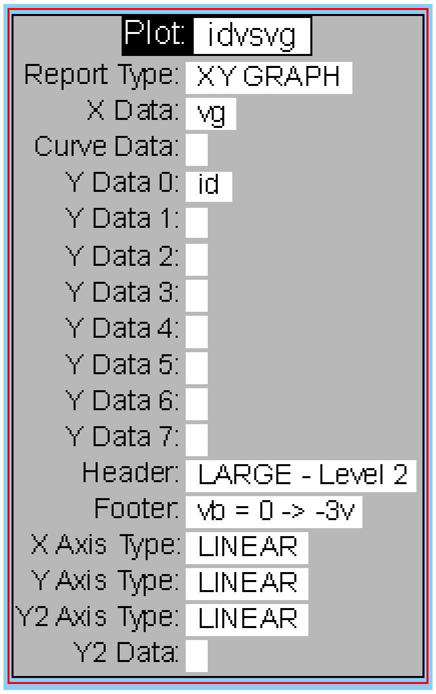
To view the measured data, click **View.**



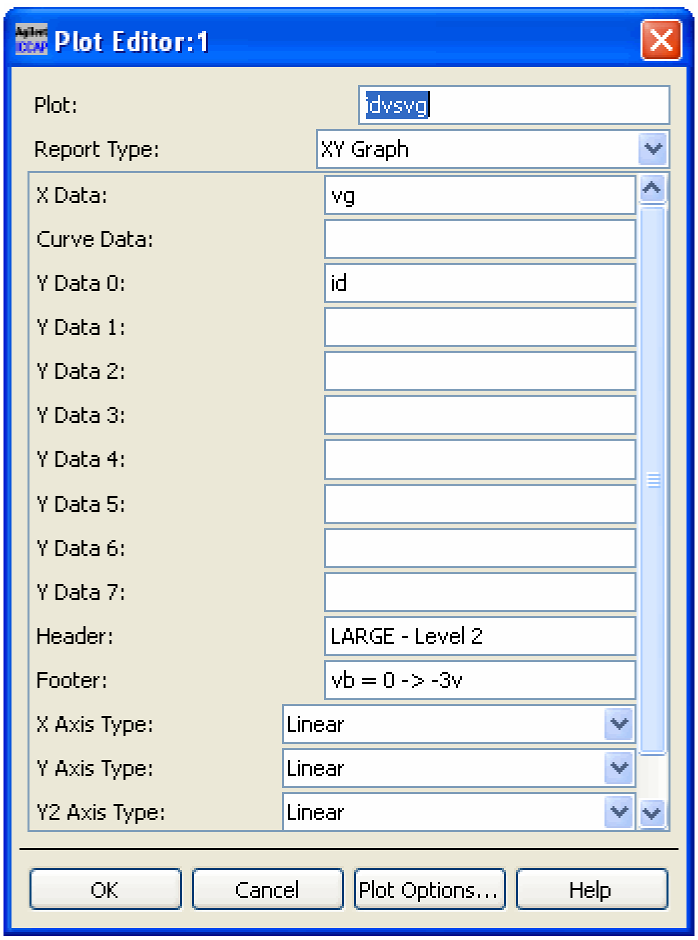
Editing the Data Table Setup

You can edit the setup for the data table through one of the following two methods:

1. **Method 1** - Edit directly in the table. Click or double-click a field and type the new data.

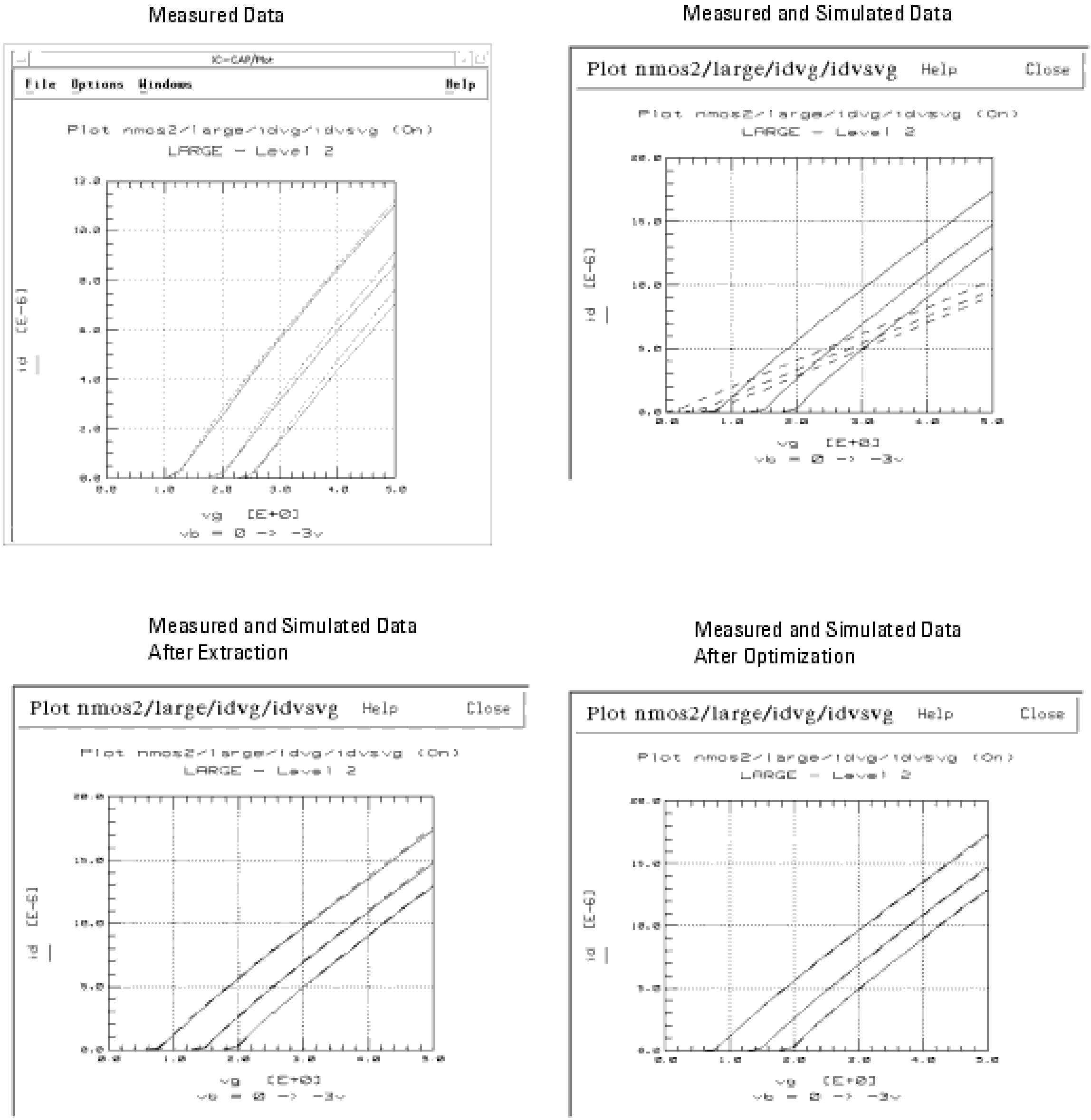
1. 

2. **Method 2** - Use the **Plot Editor**. To open the Plot Editor, select a table and then click **Edit** (or double-click on the table) to open the Plot Editor. Make changes in the respective fields and click **OK**.

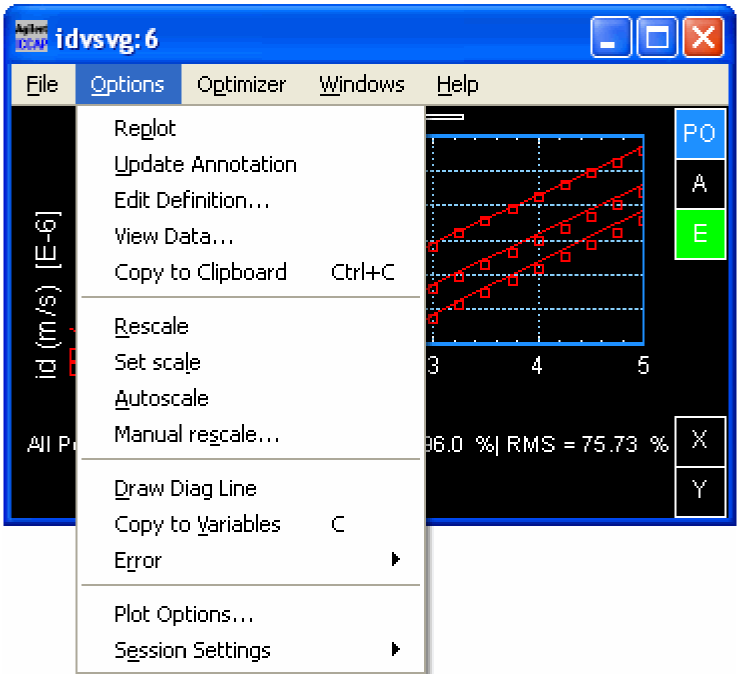


For a complete description of the Plot Editor fields, refer to *Printing and Plotting* (intro). To view the plot, click **Display Plot** or **Display All**. The IC-CAP/Plot window opens. The following figure shows examples of results.

Examples of Displayed Results



You can define display options by selecting **Options** in the Plot window. For more details, refer to *Printing and Plotting* (intro).

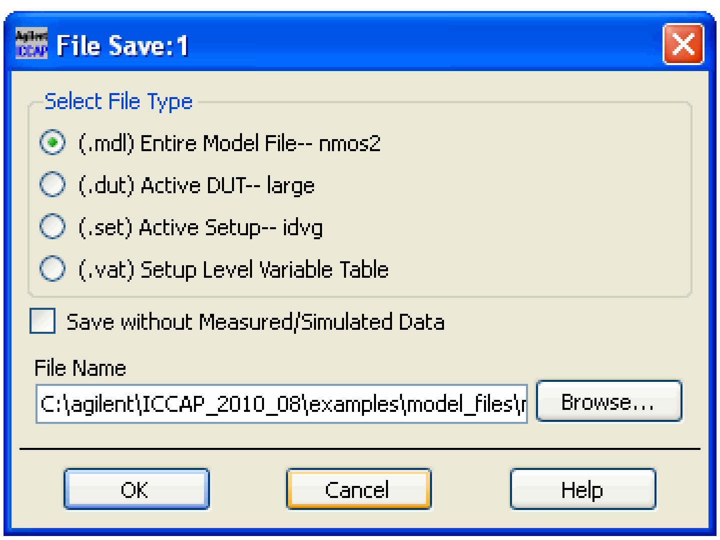


Saving the Model

When the characterization process is complete, you can save the model to a file.

To save the model to a file:

1. Select **Save**. 
2. You can save a model into various file types. Choose the **File Type** to save the model to a specific file type.



3. Type a unique name (such as, *mybjt_npn.mdl*) in the **File Name** field.
4. Click **OK** to save.

Exiting IC-CAP

To exit IC-CAP, select **File > Exit** in the IC-CAP Main window. This opens the Save As dialog box to give you an opportunity to save changes to models before exiting the program. Click on the **Exit** button to automatically close all open windows and terminate the program.

Creating and Modifying Models

In IC-CAP, all of the information required to characterize a particular device or circuit is contained in a model. With a variety of model configurations available, you can easily create new models by copying and/or modifying existing models. The changes may be structural (such as the addition or removal of setups) or may involve modification of table values only.

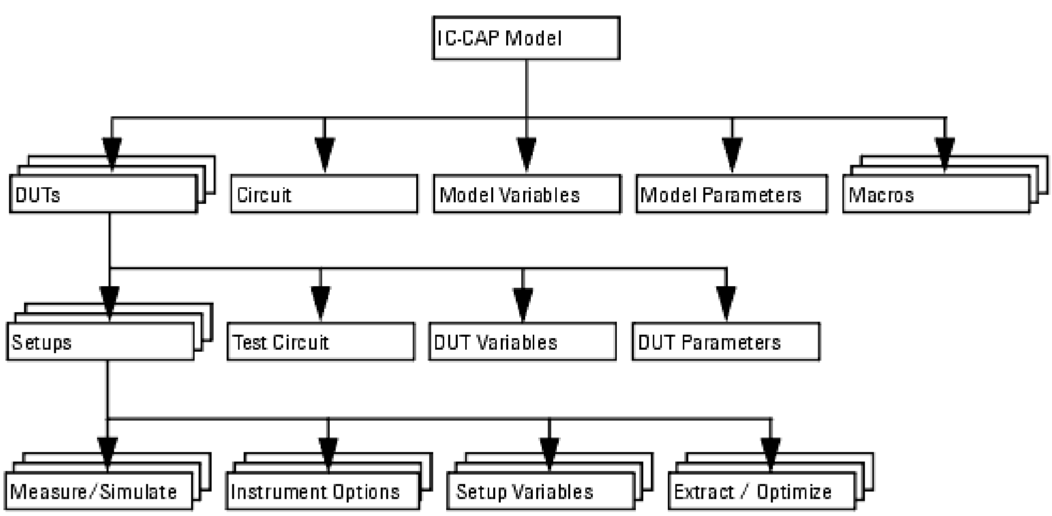
> **Note**
> Before starting this section, you must configure the necessary source and measuring instruments for the IC-CAP system. For information on configuring the system, refer to *Customization and Configuration* (custom).

Model Components

The IC-CAP model components are shown in the following figure. A model consists of these components:

- **Circuit** - Description of the circuit used to represent the actual device.
- **Model Parameters** - Values used in simulating the circuit.
- **Model Variables** - User-defined variables that are global to a model.
- **DUTs** - Representations of different physical devices or measurement configurations.
- **Macros** - Combinations of the available menu functions used to create complex operations.

IC-CAP Model Components



DUTs and Setups

Each DUT contains:

- **Test Circuit** - Description of the circuit used in a particular DUT.
- **DUT Parameters** - Values for parameters specific to a DUT.
- **DUT Variables** - User-defined variables that apply only to a DUT.

- **Setups** - Define measurements and simulations for a DUT.

Each setup contains:

- **Inputs** - A definition of the input stimuli.
- **Outputs** - A definition of the types of responses to monitor.
- **Transforms** - Specifications for different data manipulations, such as mathematical functions and extractions.
- **Plots** - Data display, in a variety of formats.
- **Setup Variables** - User-defined variables that apply only to a setup.
- **Instrument Options** - Instrument options control the state of the hardware for a setup.

Defining Model Components

When you create an IC-CAP model, you must define the components. You can create a new model and then describe its physical and electrical characteristics or you can load and modify an existing model. Some components, such as user-defined variables and test circuitry, are optional.

The general procedure for creating a model is:

- Add a model to the work area.
- Define variables that are global to the model.
- Define the device or circuit to be characterized.
- View and modify the parameter values to be used in the simulations.
- Add new (or modify existing) DUTs.
    - If necessary, define a test circuit.
    - Edit device parameters specific to the DUT
    - Define variables that are global to the DUT.
    - Add new (or modify existing) setups.
- For each setup, define the following:
    - Inputs to the device or circuit.
    - Outputs to monitor.
    - Variables global to the setup.
    - Values of instrument options for all instruments used in a setup.
    - Transforms, including mathematical functions and extractions.
    - Plots that specify graphical and tabular formats for the data.

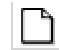
	Note For an example, see Creating a New Model.

Adding a Model to the Work Area

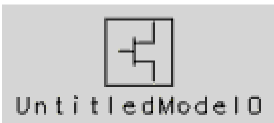
You can add a model to the work area in one of two ways:

- Open an empty template
- Open an existing model

To open an empty template:

1. In the IC-CAP/Main window, click **Create Model** on the toolbar. 

    An icon representing a new untitled model appears in the IC-CAP/Main window.



2. To open the Model window, double-click the model icon.

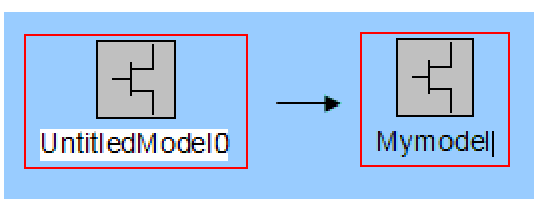
To open an existing model:

1. In the IC-CAP/Main window:
    - Click **Open** to open a model in the current directory 
    - Click **Examples** to open a model provided with the program 
2. In the dialog box, double-click **model_files** to see the list of model directories, or double-click **demo_features** to see the list of demonstration directories.
3. Double-click on a directory name, then click on a model name in the Files list to select a model file.
4. Choose **OK** to accept the selection and close the dialog box.

    An icon of the model file displays in the IC-CAP/Main window and the Model window opens. If the model displays as an icon only, double-click the icon to open the Model window.

Renaming a Model in the Work Area

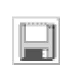
1. Click to highlight the model name in the work area.
2. Retype a new name and press **Enter**.

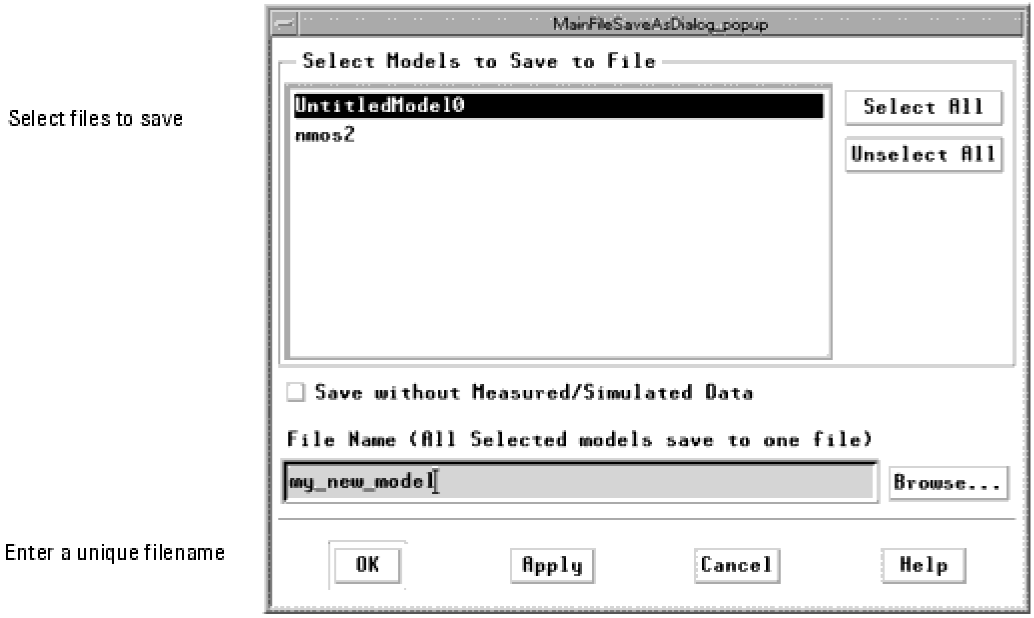


Saving Model Files

Before modifying a new model, you can save the model file to different name.

To save model data from the IC-CAP/Main window:

1. Select a model icon in the IC-CAP/Main window, then select **Save As**. 
2. Select the open models you want to save.



3. In the File Name field, enter a unique filename and click **OK**. The new model name appears in the directory list.

    If a file was saved previously by that name, an information dialog box opens.

    - To replace the existing model by that name, choose **Yes**.
    - To avoid replacing the existing model by that name, choose **No** and enter a new file name to save the model to different name.

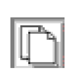
    You can use the same name as the supplied file, as long as you save the model to a different directory. For details on changing directory paths, refer to *Changing Directories* (intro).

Copying Parts of a Model

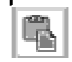
You can create a set of models, or parts of a model, that are similar to each other by using the Copy command. You can copy any part of the model that you can select. For example:

- To copy the circuit definition, parameter set, or variable table, open the Circuit, Model Parameters, or Model Variables tab.
- To copy an entire DUT, select the DUT.
- To copy an entire setup, select the Setup.
- To copy an input, output, transform, or other table in a setup, select the table (by clicking on the outside ruling) or tab.

After copying, you can place the copied parts into the new model and modify the copied parts.
To copy parts of an existing model:

1. Open the model and select the parts you want to copy.
2. Select **Edit > Copy [part type]** or click **Copy** on the toolbar. 

To place the copied parts in the new model:

1. Place the cursor in the folder where you want to put the copied parts.
2. Select **Edit > Paste [part type]** or click **Paste** on the toolbar. 

> **Note**
> To copy an entire DUT or Setup in a model, right-click on the DUT or Setup you want to copy to view the pop-up menu, then select **Edit > Copy**. To paste a Setup into another DUT, right-click on the DUT, then select **Edit > Paste**. To paste a copied DUT into another model use the directions in the procedure above.

Specifying the Circuit Definition

The circuit definition specifies the parameters to be extracted and the representation of the model in the simulator. The parameter values act as defaults in IC-CAP. In the supplied model files, these values are usually set to SPICE defaults. However, you may want to specify values for the devices or circuits that you plan to characterize, since extractions and optimizations run faster if starting parameter values are closer to the final values. For a complete description of the types of circuits that can be defined, refer to *Simulating* (sim).

Example

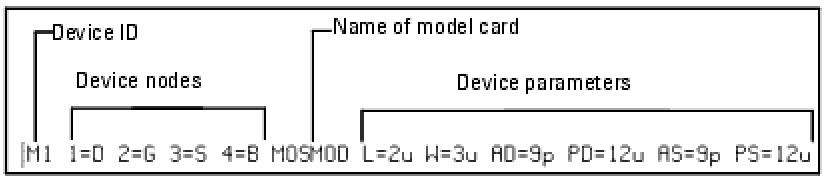
The example circuit is a supplied model, *nmos2.mdl*, that extracts parameters for the Level 2 N-channel UCB MOSFET transistor.

```
.OPTIONS ucb
M1 1=D 2=G 3=S 4=B MOSMOD L=2u W=3u AD=9p PD=12u AS=9p PS=12u
.MODEL MOSMOD NMOS
+ LEVEL = 2
+ UO = 600
+ VTO = 0
+ NFS = 0
+ TOX = 100n
+ NSUB = 1e15
+ UCRIT = 10.00K
+ UEXP = 0
+ VMAX = 1MEG
+ RS = 0
+ RD = 0
+ XJ = 0
+ LD = 0
+ DELTA = 0
+ NEFF = 1.00
+ NSS = 0
+ CGSO = 0
+ CGDO = 0
+ CGBO = 0
+ CBD = 0
+ CBS = 0
+ CJ = 0
+ MJ = 0.5
+ CJSW = 0
```

```
+ MJSW = 0.33
+ IS = 1.0E-14
+ PB = 0.8
+ FC = 0.5
```

The circuit definition begins with the Options definition (.OPTION). The definition lists the device to be measured, the device nodes, the name of the model card that contains the model parameters used by the device, and the device parameters. In the example:

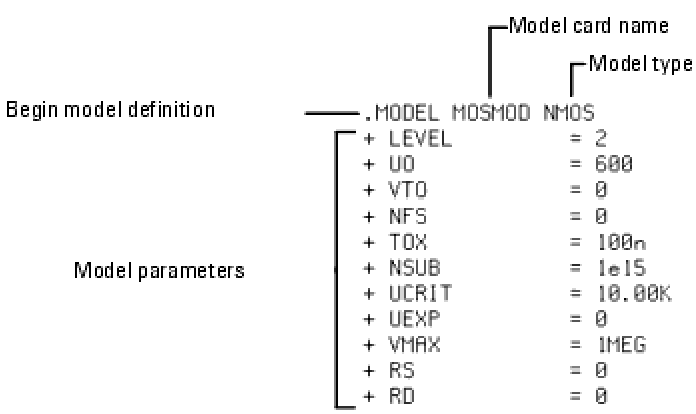
Circuit Options definition



- The first item identifies the device to be measured, M1. The character *M* indicates that the device is a MOSFET.
- Next, the device nodes are identified (1 = drain, 2 = gate, 3 = source, 4 = bulk (substrate).
- The next item, MOSMOD, is the name of the model card that contains the model parameters used by the M1 device.
- The remaining items in the Options definition specify device parameters. For real devices, refer to the design specification for the values of device parameters.

The next section is the model card definition (.MODEL). The definition lists the name of the model card, the model type, and the contents of the model card. The contents of the model card consists of the parameters that are to be extracted (model parameters) and their assigned initial values. In the example:

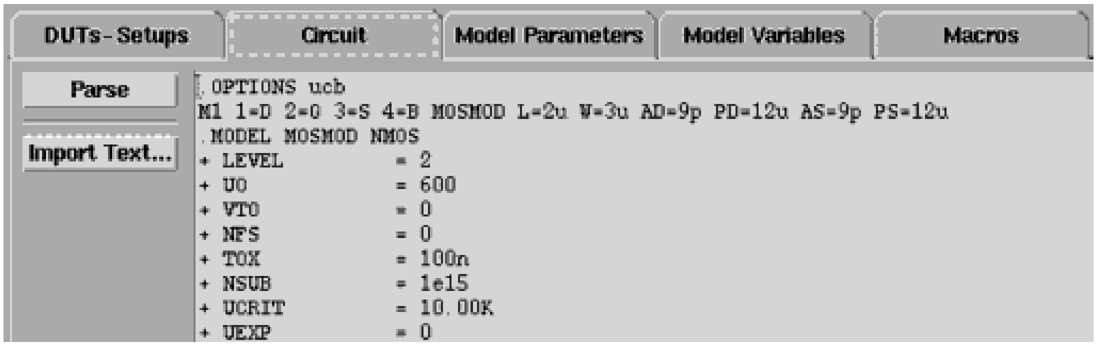
Circuit Model definition



- The name of the model card is identified (MOSMOD), then the model type N-channel (NMOS).
- The remaining lines are the contents of the model card. These consist of the parameters that are to be extracted (model parameters) and their assigned initial values. The leading + indicates that each parameter follows the preceding just as if all the parameters had been entered on a single line.

To specify the circuit definition:

1. In the Model window, click **Circuit** to open the folder.



2. Edit the existing values by selecting the value and typing over it.
3. When you move the mouse pointer out of the window, the new circuit description is parsed automatically. This parse only occurs after a change has been made to the contents of the circuit description.

Syntax errors are reported in the IC-CAP/Status window and must be corrected. For example, if the wrong type of simulator is selected and syntax errors result from the parse, you must parse the circuit after the correct simulator is specified.

To parse the circuit definition manually, click **Parse**.

> **Note**
> You can copy the circuit definition from an existing model. Open the model you want to copy. Open the Circuit folder. Select **Edit > Copy Circuit**. In the new model, select **Edit > Paste Circuit**.

Advanced Netlist Features

In order to populate the Model Parameter Set and properly netlist the circuit at simulation time, IC-CAP must 'Parse' the entered circuit to identify external nodes, the model name, and all the parameters. IC-CAP can parse general spice syntax, Agilent ADS Simulator syntax, SABER syntax and SPECTRE syntax. The parser used is dependent on the template name used in usersimulators or optional open parser interface parsing module. See *Linking a Simulator to IC-CAP* (sim).

If using a keyword or netlist syntax that is beyond the scope of the parser built into IC-CAP, it will be unable to properly read the netlist syntax and will issue a parsing error even though the netlist could be sent to the simulator being used and the simulator would accept it. It is possible to send parts of your netlist directly to the simulator by 'hiding' them from the IC-CAP parser with the keyword **#echo**. Any text following the keyword **#echo** will be ignored and copied to the output netlist unaltered apart from three keywords to be discussed later. Suppose the newer *.PARAM* HSPICE keyword is not recognized by the general spice parser in IC-CAP, this line can still be sent to HSPICE using #echo

Example

```
#echo .PARAM xyx=4
```

In addition, to **#echo**, three keywords may be used behind a #echo line to create parameters and one keyword may be used to substitute the text value of a named variable from a variable table if it exists. When using these keywords, the keywords are substuted with the parameter value from the parameter table or variable value from a variable table before the netlist is sent to the simulator. The three keywords that create parameters are **$mpar**, **$dpar**, and **$xpar** and the keyword that can substitute text from a variable if it already exists is **$var**. The syntax is the same for all and includes a parameter/variable name and default value assignment We can extend the previous example to use **$mpar**.

Example

```
#echo .PARAM xyx=$mpar(xyx=4)
```

When IC-CAP reads this line it will send everything after the #echo to the simulator **except** the $mpar(xyz=4) portion. That portion will be completely substituted with the *value* of the model parameter xyz which will be created as a result of the **$mpar** keyword. If the user had set the model parameter *xyz* to 12, then when the netlist is created for simulation, the simulator would see:

Example When Simulated

```
.PARAM xyz=12
```

The difference between **$mpar**, **$dpar**, and **$xpar** is as follows:

- **$mpar** If used in the model circuit, these parameters are created in the Model Parameters table of the model. If used in a DUT level Test Circuit, they are treated as device parameters that will appear in Device Parameters of that DUT only.
- **$dpar** If used in the model circuit, these parameters will appear in **all** DUT Device Parameter tables. Apart from that they will be treated as regular parameters. If used in a DUT level Test Circuit, they will act just like **$mpar** parameters and will appear in that DUT only.
- **$xpar** These parameters behave like **$dpar** described above, with one key difference. When IC-CAP netlists the device instance line for the simulation, **$xpar** parameters will be netlisted along with this line to be passed to the model or circuit to be simulated. Consider the simple diode circuit:

Example

```
D1  1 = A  2 = C  DIODE AREA=1
.MODEL DIODE D IS = 1E-14 N = 1.0
```

When simulated, the above netlist will be sent to spice2 as:

Example When Simulated

```
Simulation Input File
.options
+ TNOM = 27
.MODEL diode D
+ IS = 1.001E-014
+ N = 0.9997
DCKT 1 2 diode
+ AREA = 1.1
* START SOURCES
VAGRO 1 0 DC 0
VCGRO 2 0 DC 0
* END SOURCES
.DC VAGRO 0.3 1 0.025
.PRINT DC I(VAGRO)
.END
```

Notice that the values in the Model Parameter page for IS, N, and AREA differ from those specified in the netlist-- when simulating IC-CAP always uses the values from the Model Parameters and Device Parameters tables. The lines of the netlist up until the DCKT line are taken from the Test Circuit and Model Circuit with only the device name and nodes and parameter values being substituted for the current simulation. However, the DCKT line is completely fabricated based on the D1 line provided in the netlist. To understand the difference between **$xpar** and **$dpar** consider the netlist below when compared with the example above.

Example

```
#echo $xpar(AREA=1)
#echo * This is a comment referencing $xpar(AREA2=1)
#echo * this is just a comment noting that unused device parameter ex=$dpar(ex=12)
D1  1 = A  2 = C  DIODE
.MODEL DIODE D IS = 1E-14 N = 1.0
```

Here we do not specify *AREA* on the D1 line, but we want IC-CAP to treat *AREA* in the same way so we use **$xpar**. By contrast we create a **$dpar** parameter for the parameter *ex*. Observe how ex and AREA are treated differently when netlisted for the simulator:

Example When Simulated

```
Simulation Input File
.options
+ TNOM = 27
* This is a comment referencing
* this is just a comment noting that unused device parameter ex=13
.MODEL diode D
+ IS = 1.001E-014
+ N = 0.9997
DCKT 1 2 diode
+ AREA = 1
+ AREA2 = 1
* START SOURCES
VAGRO 1 0 DC 0
VCGRO 2 0 DC 0
* END SOURCES
```

```
.DC VAGRO 0.3 1 0.025
.PRINT DC I(VAGRO)
.END
```

AREA and *AREA2* are netlisted with the DCKT line like the previous example. This is because $xpar was used which specifies that such parameters are to be netlisted on the instance line. The parameter *ex* in contrast is defined using $dpar() which only has the simple substitution effect of placing the current value from the device parameter table (13) in its place of the text behind the **#echo**. Not only is the **$xpar** netlisted with the instance line, it is **not** substituted in the **#echo** line that defines it (see the #echo for $xpar(AREA2=1)). In fact, if no other text is specified behind the #echo apart from the **$xpar** specification, the #echo line is not sent to the netlist at all (see the **#echo** line that defines $xpar(AREA=1)).

Finally, $var differs from the previously discussed 3 keywords in three ways. First it will not create a variable-- it will only do the substitution if the variable is found. If no variable is found it will instead use the default value specified. Secondly, the substitution is always text (or string) based. That is a variable having a numeric value is never treated as a number so PARAMETER_PRECISION or WORKING_PRECISION will have on effect on this substition. The value part of the $var(name=value) may be specified with quotes $var(myvar="default value"). Finally, the variable is searched from the setup being simulated so it is possible to have each setup provide a different string substitution if each setup has the same variable defined with different values.

Example

```
#echo include "$var(path="/users/icuser/mycir.va")"
```

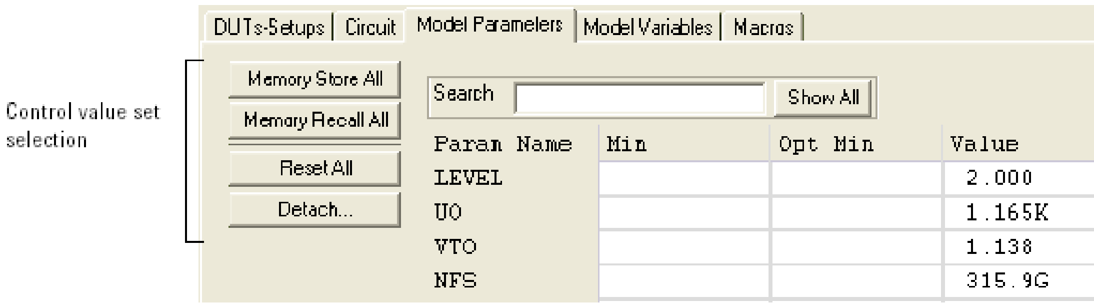
Implementing Macro Models

Macro modeling provides a powerful method for defining and simulating new models in IC-CAP. The approach is particularly useful when a complex topology is required. There are several ways to create macromodels. For example, enter the model in the form of a circuit definition and include any model component that is supported by SPICE. Another method is to enter new model equations using the dependent voltage and current sources, although the equations for these sources are limited to polynomial functions.

Specifying Model Parameters

The Model Parameters folder displays the current parameter values. These parameters are common to all DUTs for that model and are used in a simulation. When a new model is read, these values are set to the defaults defined in the circuit. The values change after extractions and optimization are performed.

To view the model parameters, select the **Model Parameters** folder.



The Parameters table contains the same parameters as those specified in the *MODEL* definition of the Circuit. However, by editing the table, the parameter values can be set independently and can be specified as a real number or an equation. In addition, you can set limits by entering minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values.

> **Note**
> The value fields can be set either to an actual number or to functions of other data available in IC-CAP. For example, if you enter RS as the value for the RD parameter, this parameter is always equal to the RS parameter.

To edit the model parameters:

1. In the Model window, select **Model Parameters**.
2. Edit the parameter value fields and select an action from the following:
   - To temporarily store the current set of parameter values, click **Memory Store All**.
   - To recall stored parameter values, click **Memory Recall All**.
   - To reset the default values specified in the circuit definition, click **Reset All**.

> **Note**
> You can copy a current set of parameter values from an existing model. Open the model you want to copy. Select the **Model Parameters** folder. Select **Edit > Copy Parameter Set**. In the new model, select **Edit > Paste Parameter Set**.

Setting Model Variables

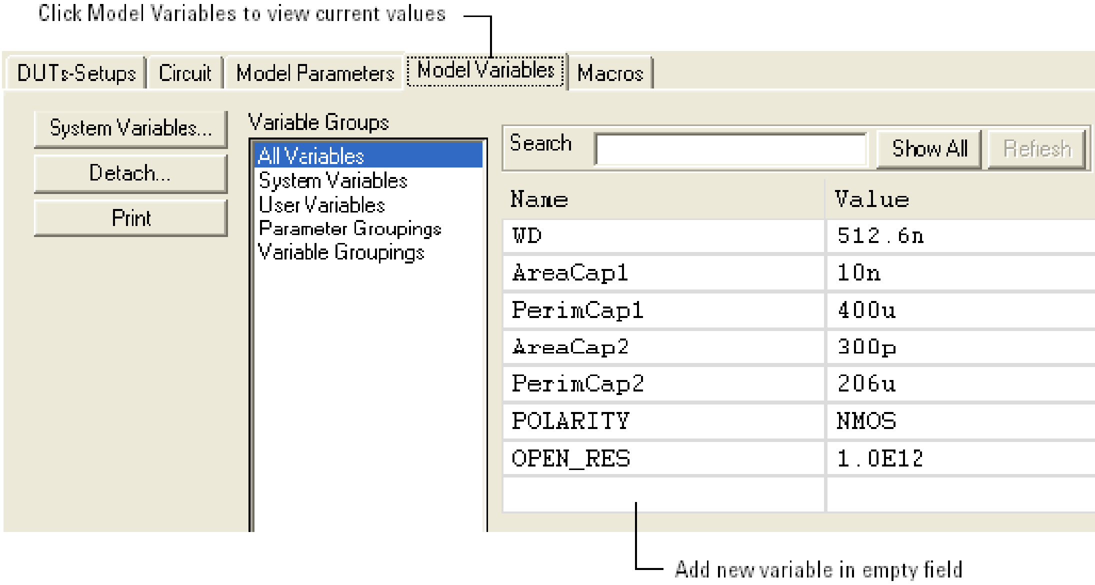
Variables can be defined at the IC-CAP system level or at the model, DUT and setup levels. You can define as many variables as you need and use these variables for other functions, such as extractions.

The Model Variables folder contains the names and values of all variables that are global to the model.

> **Note**
> Do not attempt to enter an expression when editing model variables. Expressions in variables are not evaluated when the variables are referenced. If a variable is used in a numeric expression, enter a single number only.
>
> Do not enter the name of a system variable into the value field. The value of the system variable is not evaluated when the variable is referenced.

To view or edit the model variables:

1. In the Model window, select **Model Variables**.



2. Edit the model variable fields by selecting the value and typing over it.

Note
To view system defaults from the Model Variables folder, select the  button.

To specify new model variables:

1. In the Model window, select **Model Variables**.
2. Enter the names and values for all model variables:
   - Click the first empty row in the *Name* field and type a name for the variable.
   - Press Tab to move to the *Value* field. Type a value for the variable.
   - Press Enter to accept the value and open another row.

Note
You can copy the model variables from an existing model. Open the model you want to copy. Select the **Model Variables** folder. Select **Edit > Copy Variable Table**. In the new model, select **Edit > Paste Variable Table**.

Finding Model Parameters or Variables

Use the search box on the *Model Parameters* and *Model Variables* folders to find subgroups of parameters and variables. In addition, you can save any subgroup to a name for easy retrieval using the search box.

Using the Search Rules

In the search box, special characters are: *, <space>, and <comma>.

Handling rules:

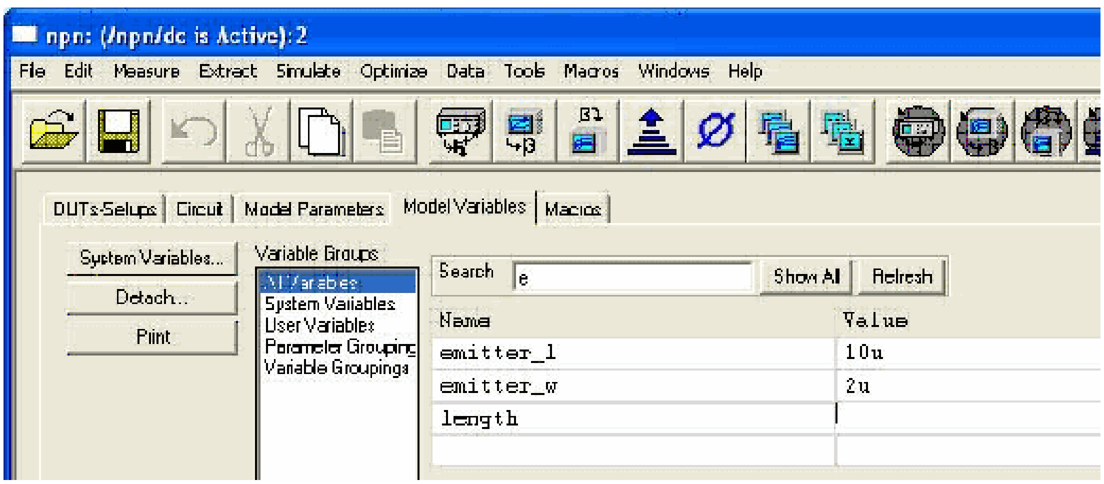
1. * means zero or more characters. An implied * will apply to the last cluster of letters.

2. <space> delimits clusters of letters but the implied * remains
3. ',' delimits clusters and terminates the implied *
4. [] is used for searching all elements in a ICCAP_ARRAY

Example	Description	Heading Rule
ABC	ABC* (implied trailing *)	1
ABC (trailing space)	ABC* (implied trailing *)	2
ABC DEF	ABC* and DEF* (implied * for both)	1,2
ABC* DEF	ABC* and DEF* (implied * on DEF)	1
ABC,	ABC (no more implied *)	3
ABC,DEF	ABC and DEF* (implied * on DEF)	1,3
ABC*, DEF	ABC* and DEF* (implied * on DEF)	1
*AB	*AB* (implied trailing *)	1
AB[]	all elements in ICCAP_ARRAY AB	4
AB[1][]	all elements in ICCAP_ARRAY AB[1]	4

Refresh button in variable page:
If a new variable is added to the current search results, the newly added variable will remain in current search results until the *Refresh* button is selected.

For example, insert a variable not begin with character *e*. In this example, *length* remains visible until the *Refresh* button is selected.



Creating Parameter Groups

Use parameter groups to select parameters.
MDL file specific groups

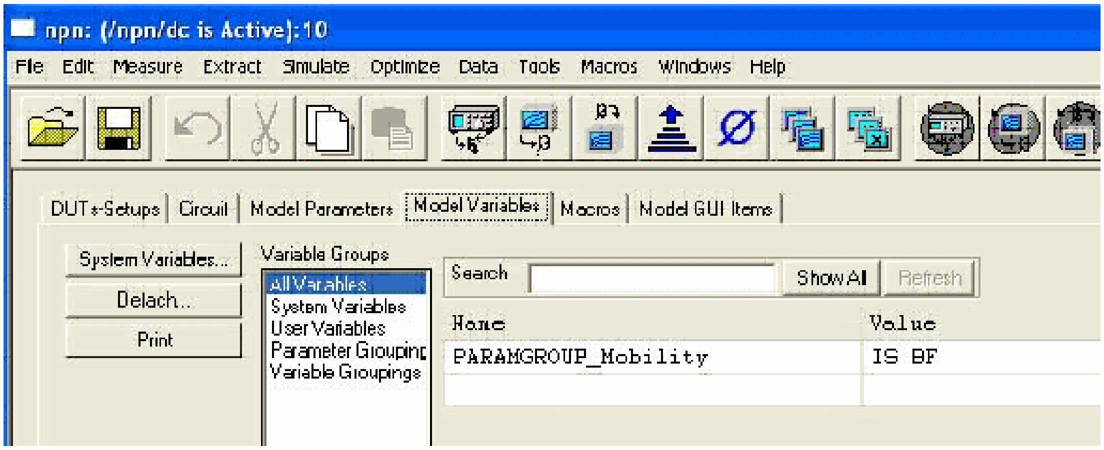
Standard Groups:

- *All Parameters* includes all parameters in the current level.
- *Ungrouped* includes parameters in *All Parameters* that are not included in other groups.

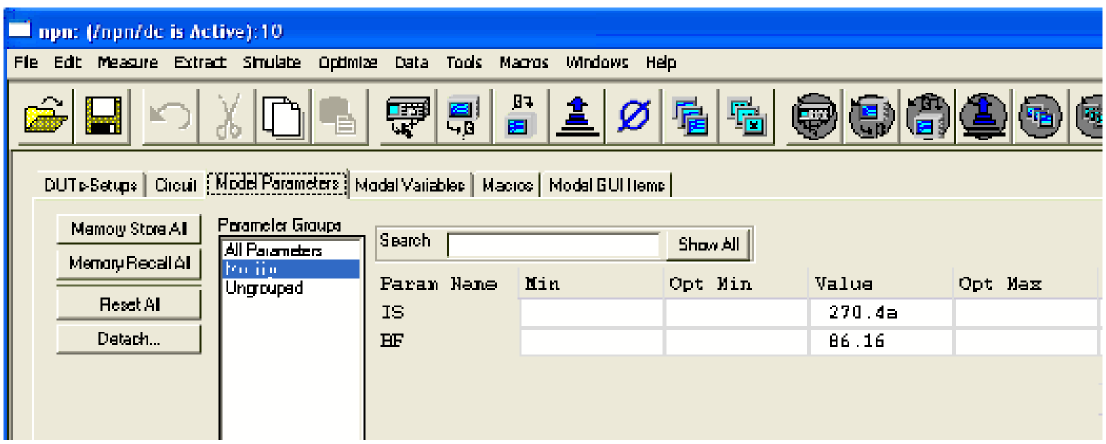
User Defined Groups:

Use the keyword *PARAMGROUP_* to define a parameter group. Specify the parameters in the value column, using <space> as the delimiter. Group name can be nested in the *Value* column. If there are no user defined groups, the list will not be displayed in the *Model Parameters* folder.

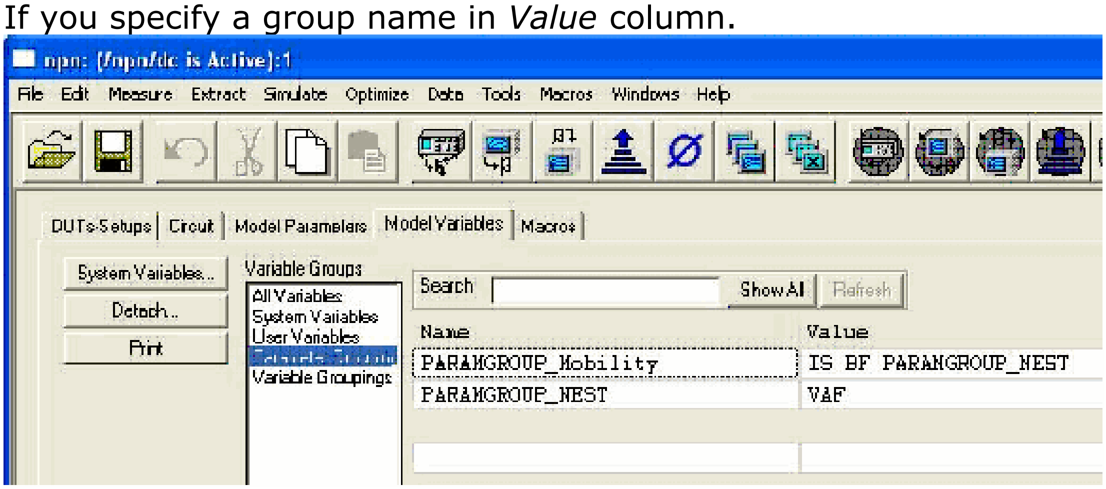
For example, define a variable that begins with the keyword *PARAMGROUP_* as shown below.



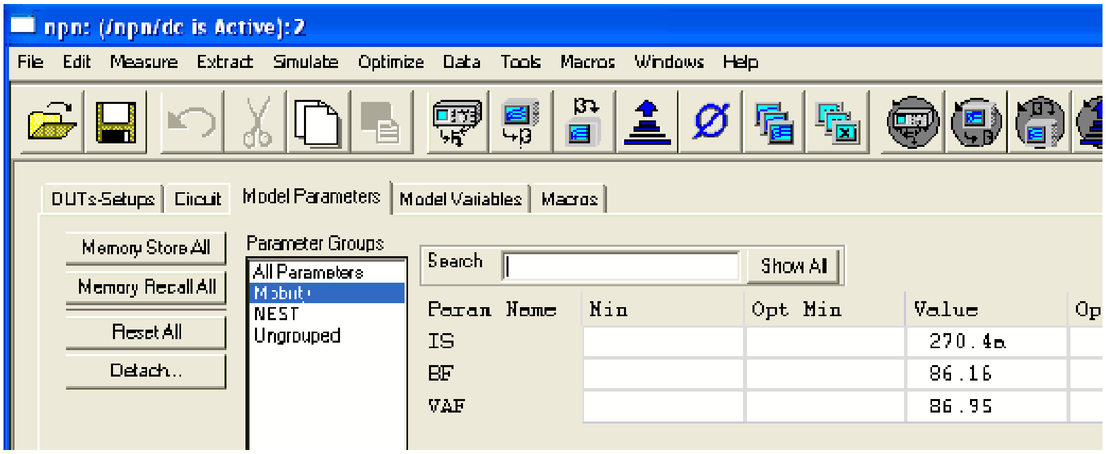
Then the parameters included in the *Value* column are displayed together in the *Model Parameters* folder as shown below.



If you specify a group name in *Value* column.



It is displayed it in the *Model Parameters* folder as shown below.



Builtin Groups

The user defined groups for parameters can also be defined in a configuration file. Format of the file:

- [] is used to specify technology name.
- # at the beginning means this line is a comment.
- \ at the end of a line means the following line is also included in current group. device. means the group is defined for the device level, no prefix means for the model level.
- A empty line will be ignored.

For example,

```
# Builtin Config File

[BSIM3]
Flags=LEVEL BINUNIT MOBMOD \
   CAPMOD NOIMOD \
   PARAMCHK

Process=DELTA TNOM TOX \
   TOXM NCH XJ NGATE RSH

[BSIM4]
Flags=LEVEL
```

Steps for configuration:

1. Specifying file location in *iccap.cfg*.
   For example:
   `ICCAP_BUILTIN_GROUPS_TEXT={$ICCAP_ROOT}/config/builtin_groups.txt`
2. Select the technology name in the variable table.
   For example:
   `BUILTIN_TECHNOLOGY_NAME=BSIM3`

Creating Variable Groups

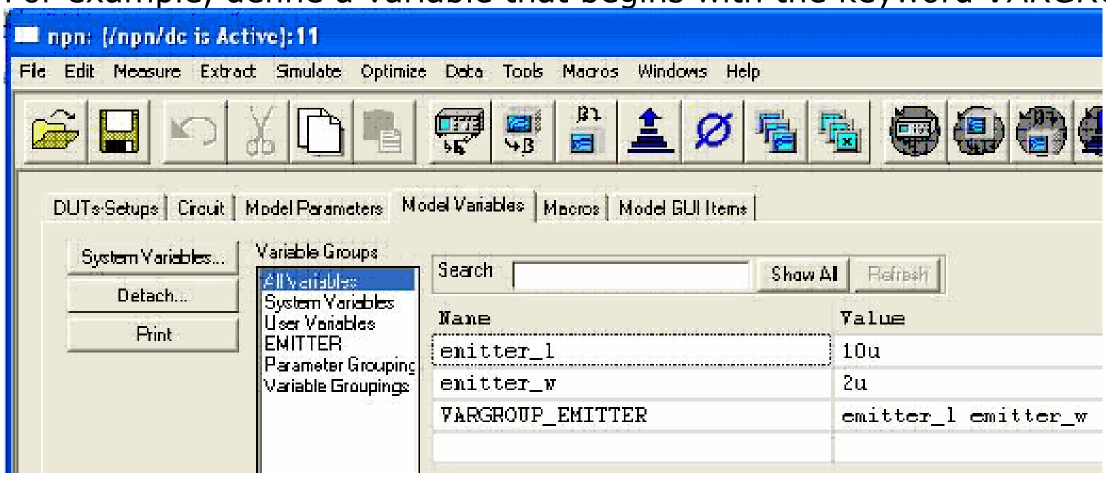
Variable groups are used for selecting variables.

Standard Groups:

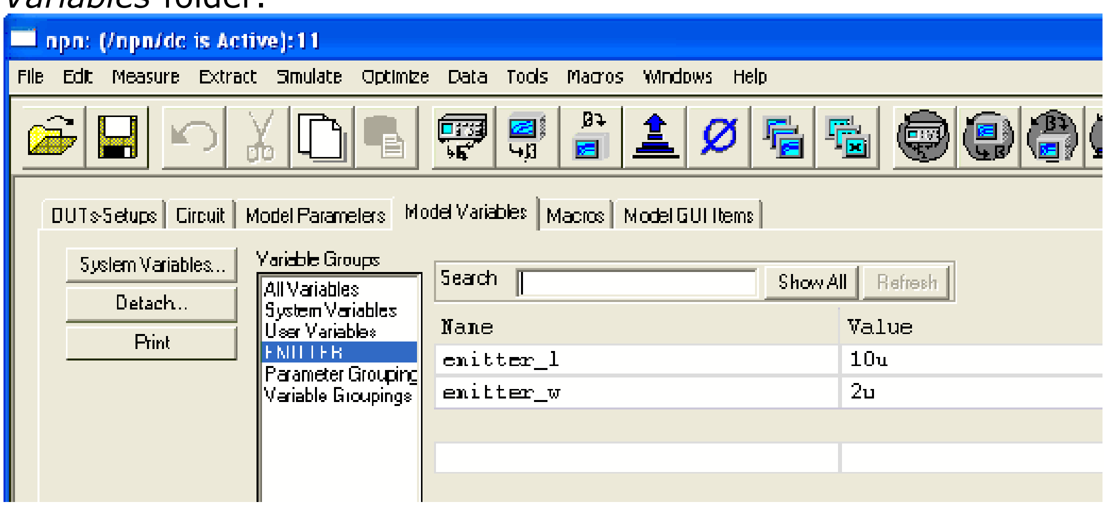
- *All Variables* includes all variables in current level.
- *System Variables* includes all system variables defined in the current level.
- *User Variables* includes variables in *All Variables* that are not included in other groups.
- *Parameter Groupings* includes variables that begin with keyword *PARAMGROUP_*.
- *Variable Groupings* includes variables that begin with keyword *VARGROUP_*.

User Defined Groups:
A variable group can be defined in the *Model Variables* folder with the keyword *VARGROUP_*. Specify the variables in the *Value* column, using <space> as delimiter. Group name can be nested in the *Value* column.

For example, define a variable that begins with the keyword *VARGROUP_* as shown below.



Then the variables included in the *Value* column are displayed together in the *Model Variables* folder.

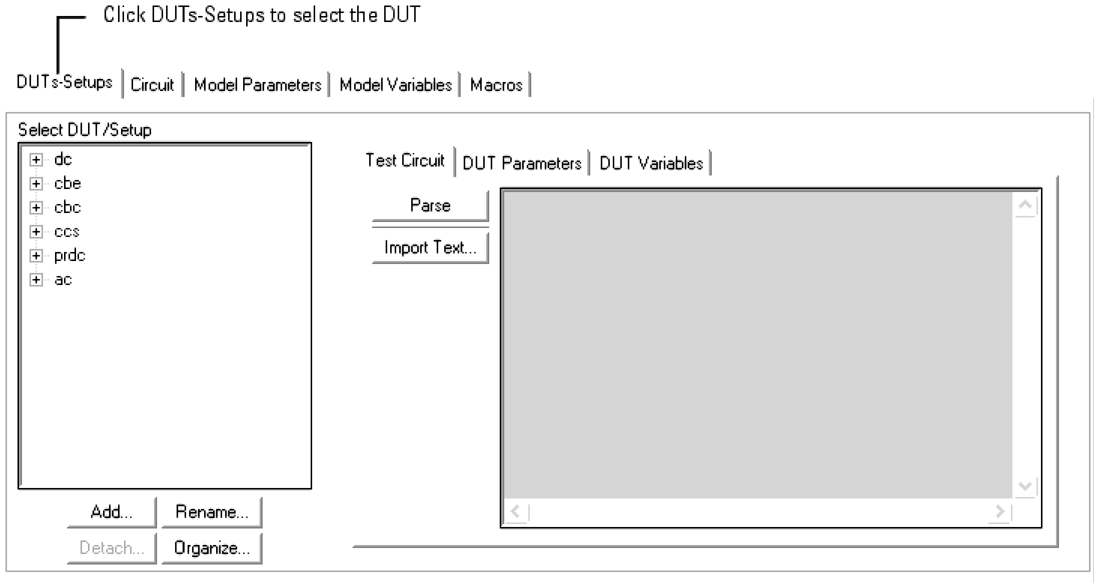


Defining DUT Options

IC-CAP models usually contain several DUTs. DUTs contain groups of setups that have a similar physical connection to the device. In addition to setups, each DUT contains its own variables, parameters and test circuit. If two setups require differences in either of these areas, you must define different DUTs for each.

Before you can measure a device, you must make any needed changes to the DUT options.

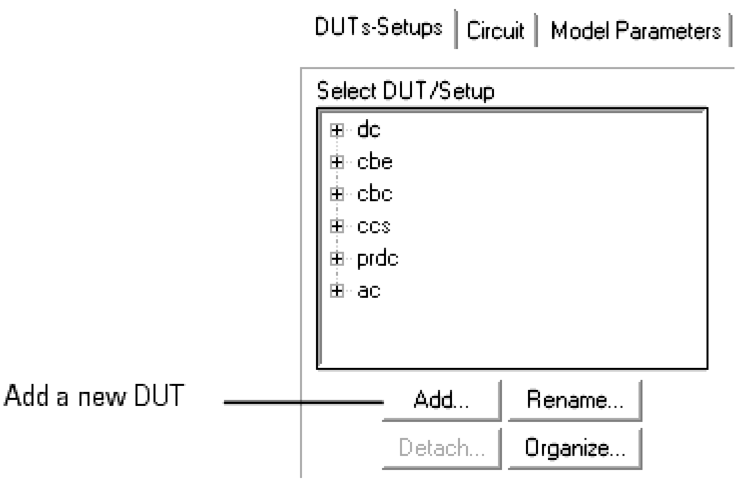
To view the DUT options and make a device active, select the **DUTs-Setups** folder. Then select the DUT.



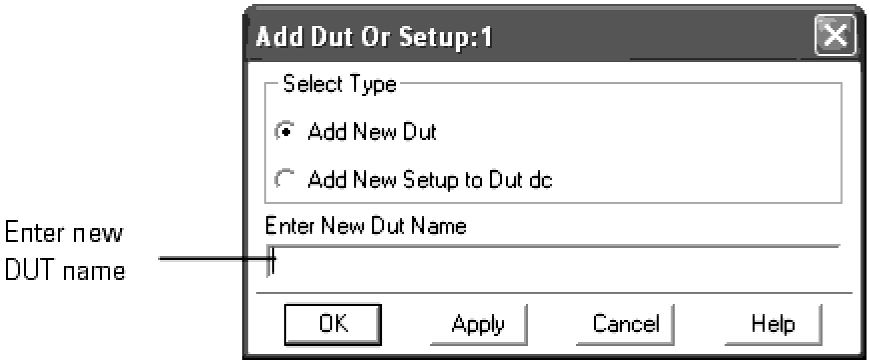
Adding a DUT

To add a DUT:

1. In the DUTs-Setups folder, click **Add**.



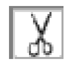
2. In the dialog box that opens, select **Add New DUT**.
3. In the Enter New DUT Name field, type a name for the new DUT and click **OK**.



See also: Organizing DUTs

Deleting a DUT

To delete a DUT:

1. In the DUTs-Setups folder, select the DUT you want to delete.
2. Select **Edit > Cut DUT**, or click **Cut** on the toolbar. 

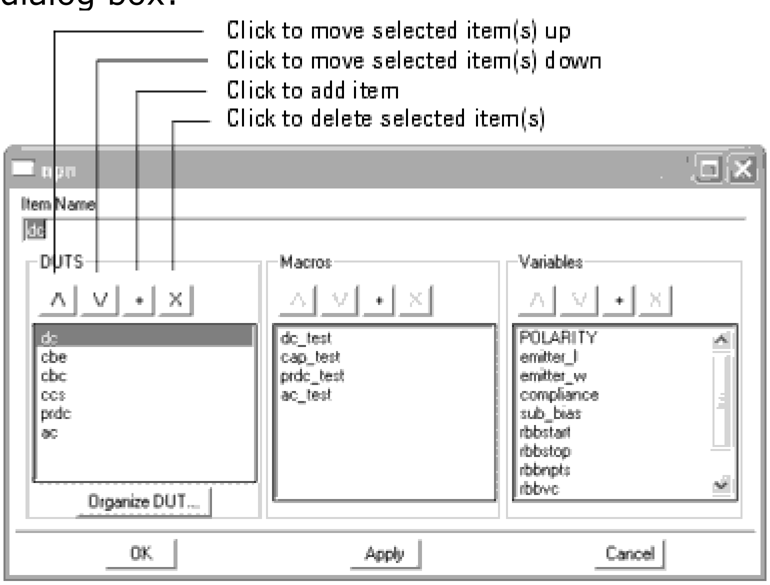
> **Note**
> You can also delete a DUT by right-clicking on the DUT to view its pop-up menu. Then select **Edit > Cut**.

See also: Organizing DUTs

Organizing DUTs

To move a DUT in the list:

1. Select **Tools** > **Organize Model** from the Model window to display the following dialog box:



1. From this dialog box, select one or more DUTs.
2. Above the selected DUT(s), click the move up icon to move the selected DUT(s) up in the list, or click the move down icon to move the selected DUT(s) down in the list. Each click moves the selected DUT(s) one position.

To add a DUT:

1. Choose where you want to add the DUT in the list:
   - Select a DUT if you want to add a DUT after that position.
   - Do not select a DUT if you want to add a DUT to the first position.
2. Click the plus icon.
3. A new DUT named *Untitled* is added. Type the desired name in the *Item Name* field then select the Enter key.

To delete DUTs:

1. Select one or more DUTs.
2. Click the delete icon.

To apply you changes and leave the dialog box open, select the **Apply** button.

To apply your changes and close the dialog box, select the **OK** button.

To close the dialog box without making changes, click the **Cancel** button.

See also: *Organizing DUTs and Setups* (intro)

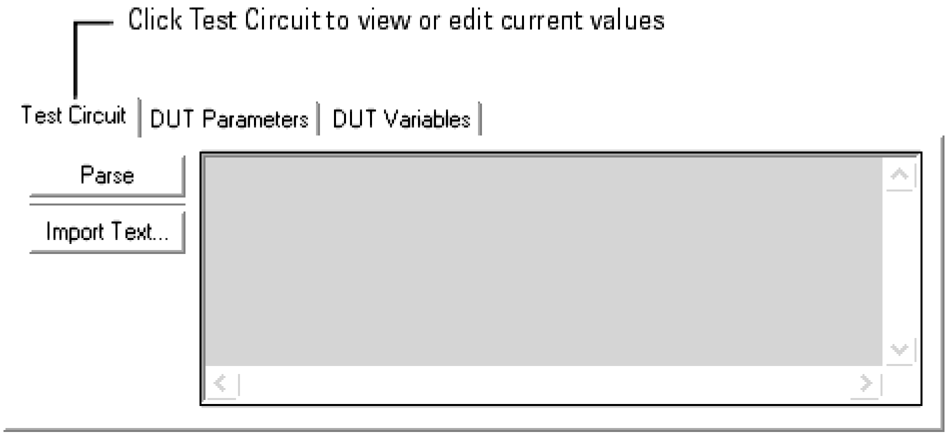
Specifying a Test Circuit

Typically, the test circuit is empty. However, you can use the Test Circuit folder to attach additional circuitry between the device or circuit being modeled and the measurement hardware. For example, an OpAmp model may need to be connected in different configurations for different measurements.

When a test circuit is defined for a DUT, the parameters in the test circuit appear in the DUT parameters folder. For complete information on defining Test Circuits, refer to *Simulating* (sim).

To specify a test circuit:

1. In the DUTs-Setups folder, select the DUT. Then select **Test Circuit**.



2. Edit the existing test circuit by typing over the existing values or enter new test circuit data in the window.

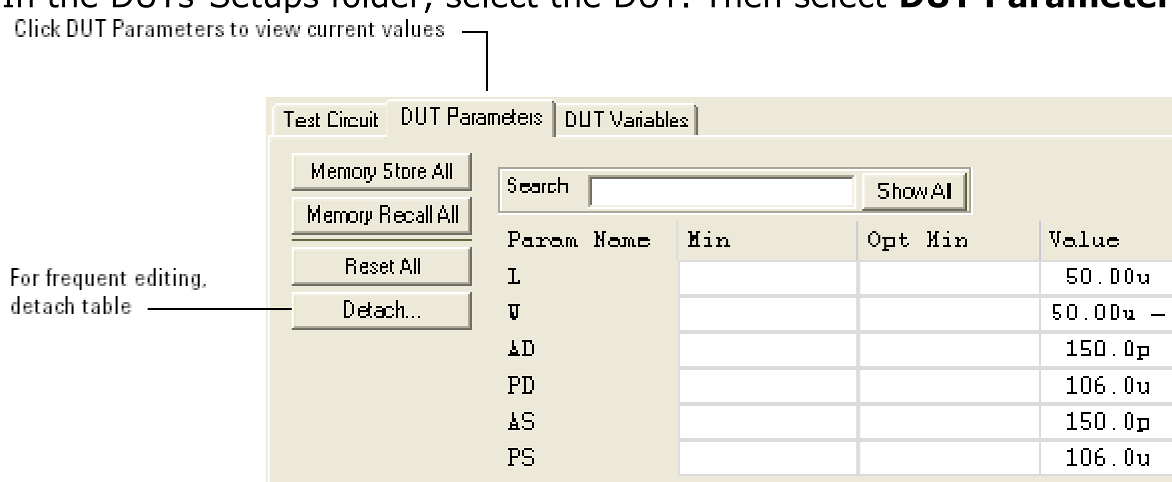
> **Note**
> You can copy a test circuit from an existing model. Open the model and select the DUT you want to copy. Select the **Test Circuit** folder. Select **Edit > Copy DUT/Test Circuit**. In the selection dialog box, choose Selected **Test Circuit**. Then select **Edit > Copy**. In the new Model window, select the **Test Circuit** folder. Select **Edit > Paste Test Circuit**.

Defining DUT Parameters

The DUT parameters may take on different values in each DUT. For example, the DC characterization of the MOSFET requires measurements on three DUTs (*large, narrow,* and *short).* Each of these DUTs have different specifications for channel length (L) and width (W). You can specify these different values in the DUT Parameters folder. In addition, you can set limits by defining minimum and maximum values for each parameter. If a parameter value is outside its limits, it will be clamped to its minimum or maximum value.

To specify DUT parameters:

1. In the DUTs-Setups folder, select the DUT. Then select **DUT Parameters**.



2. Select the **Param Name Value** field and edit the DUT parameters for the device being measured.
Optionally, select the **Param Name Min** field and set minimum values for the DUT

3. parameters.
4. Optionally, select the **Param Name Max** field and set maximum values for the DUT parameters.
5. Select an action from the following:
   - To temporarily store the current set of parameter values, click **Memory Store All**.
   - To recall stored parameter values, click **Memory Recall All**.
   - To reset the default values specified in the circuit definition, click **Reset All**.

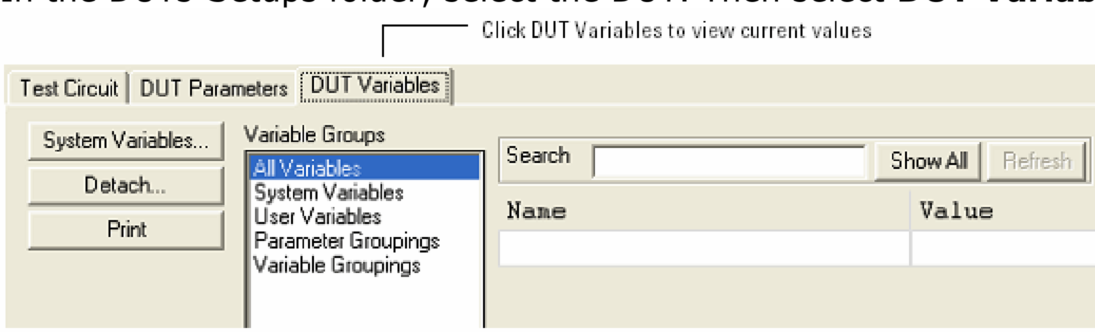
> **Note**
> You can copy DUT parameters from an existing model. Open the model and select the DUT you want to copy. Select the **DUT Parameters** folder. Select **Edit > Copy DUT/Device Parameter Set.** In the selection dialog box, choose **Selected Device Parameter Set**. In the new Model window, select the **DUT Parameters** folder. Select **Edit > Paste Device Parameter Set**.

Defining DUT Variables

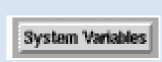
You can define variables that are accessible only to the DUT or its components in the DUT Variables folder. After you have defined the variables, you can reuse these definitions by applying the *Import* and *Export* commands.

To specify new DUT variables:

1. In the DUTs-Setups folder, select the DUT. Then select **DUT Variables**.



2. Enter the names and values for all DUT variables:
   - Click the first empty row in the Name field and type a name for the variable.
   - Press Tab to move to the Value field. Type a value for the variable.
   - Press Enter to accept the value and open another row.

> **Note**
> To view system defaults from the DUT Variables folder, select the  button.

> **Note**
> You can copy DUT variables from an existing model. Open the model and select the DUT you want to copy. Select the **DUT Variables** folder. Select **Edit > Copy DUT/ Variable Table**. In the selection dialog box, choose **Selected Variable Table**. In the new Model window, select the **DUT Variables** folder. Select **Edit > Paste Variable Table**.

Defining Setup Options

You can define setup options from the DUTs-Setups folder. A setup contains folders for defining measurement and simulation options, instrument options, setup variables, transforms, and results.

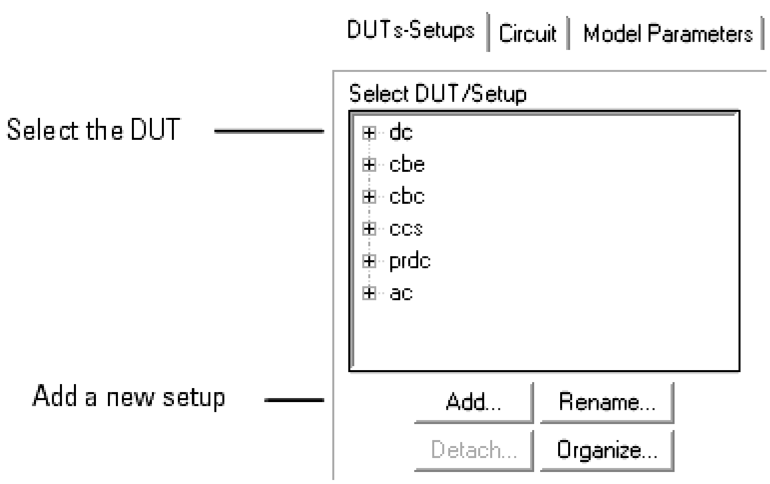
The combination of setup options you specify depends on the following:

- Measurement or simulation to perform. For example, to perform the measurement and simulation, you can specify four input voltages and one output current.
- Extractions or data manipulations required for characterization. For example, to extract values for specific model parameters and optimize the parameter values to achieve the best possible fit between measured and simulated data, you can add a transform that performs the extraction and a transform that performs the optimization.
- Desired data presentation. For example, to view the data, and compare the measured and simulated results, you can add a plot to the setup.

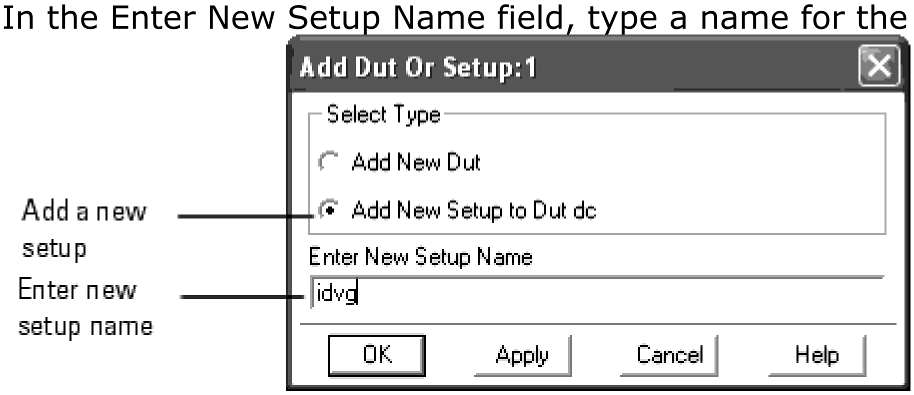
Adding a New Setup

To add a new setup to a DUT:

1. In the DUTs-Setups folder, select the DUT then click **Add**.



2. In the dialog box:
   - Select **Add New Setup to DUT [selected]**.
   - In the Enter New Setup Name field, type a name for the new setup.



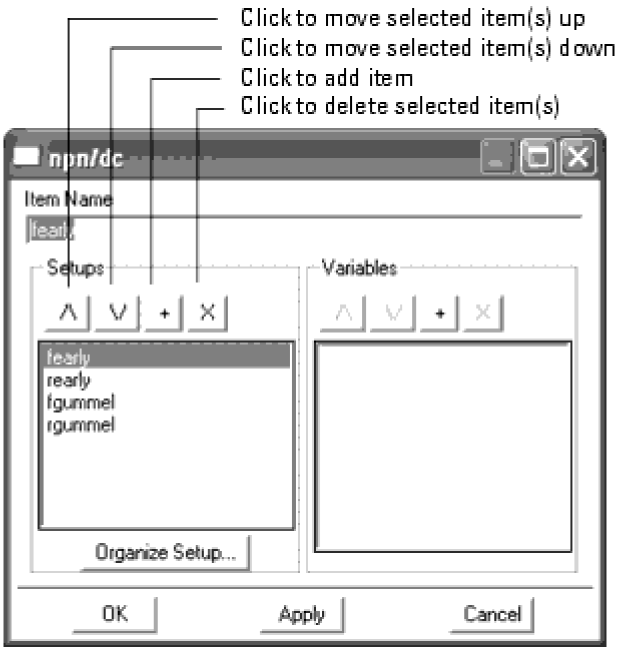
3. Click **OK**. The new setup is added to the DUT.

See Also: Organizing Setups

Organizing Setups

To move a Setup:

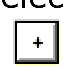
1. In the DUTs-Setups folder, select the DUT then click **Organize** to display the following dialog box:



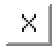
> **Note**
> You can also access this dialog box by first selecting *Tools* > *Organize Model* from the Model window, then select the *Organize DUT* button.

2. In the dialog box, select one or more Setups.
3. Above the selected Setup(s), click the move up  icon to move the selected Setup(s) up in the list, or click the move down  icon to move the selected Setup(s) down in the list. Each click moves the selected Setup(s) one position.

To add a Setup:

1. Choose where you want to add the Setup in the list:
   - Select a Setup if you want to add a Setup after that position.
   - Do not select a Setup if you want to add a Setup to the first position.
2. Click the plus  icon.
3. A new Setup named *Untitled* is added. Type the desired name in the *Item Name* field then select the Enter key.

To delete a Setup:

1. Select one or more Setups.
2. Click the delete  icon.

To apply you changes and leave the dialog box open, select the **Apply** button.

To apply your changes and close the dialog box, select the **OK** button.

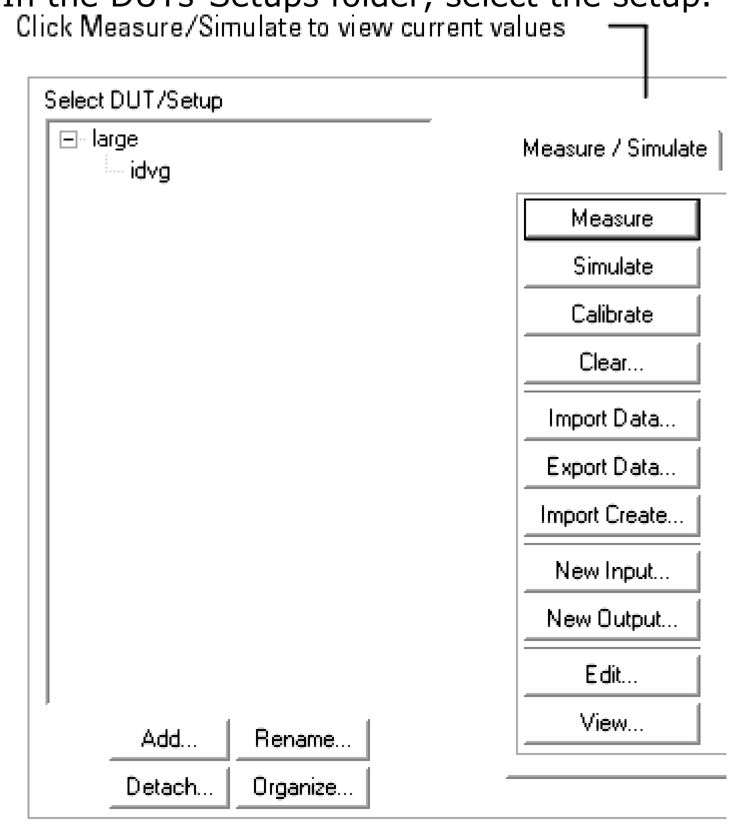
To close the dialog box without making changes, click the **Cancel** button.

See also: *Organizing DUTs and Setups* (intro)

Specifying Measurement and Simulation Options

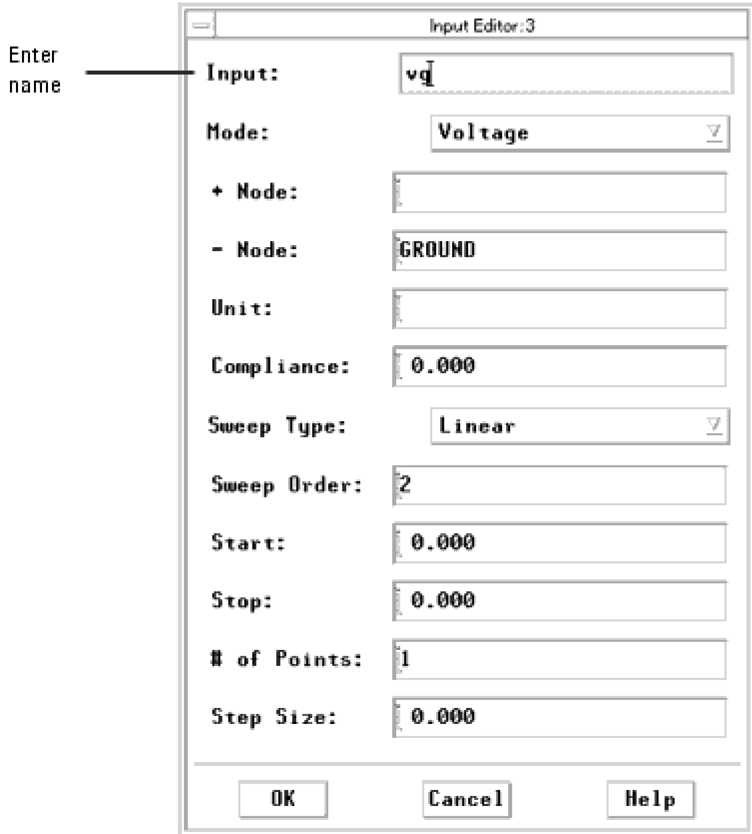
You can specify measurement and simulation options either by adding new inputs and outputs or by modifying existing values.
To add inputs:

1. In the DUTs-Setups folder, select the setup. Then select **Measure/Simulate**.

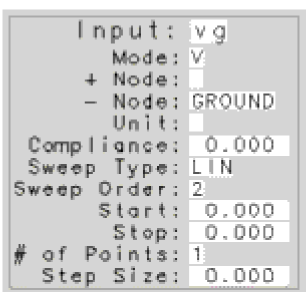


2. Click **New Input** to open the Input Editor.

3. In the Input field, enter a name for the new input.



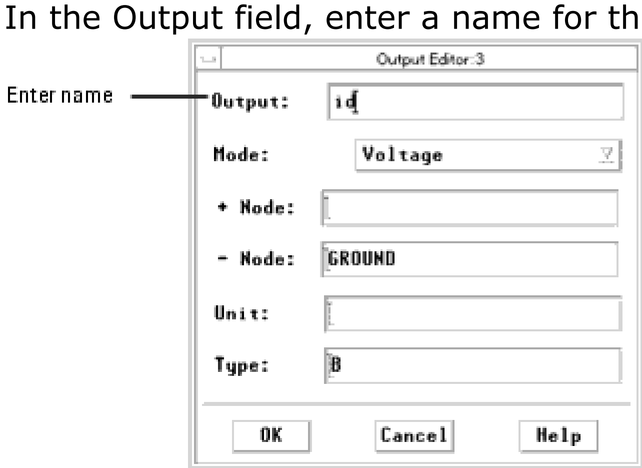
4. Click **OK** to add the input to the Measure/Simulate folder.



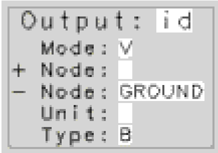
To add outputs:

1. Click **New Output** to open the Output Editor.

2. In the Output field, enter a name for the new output.

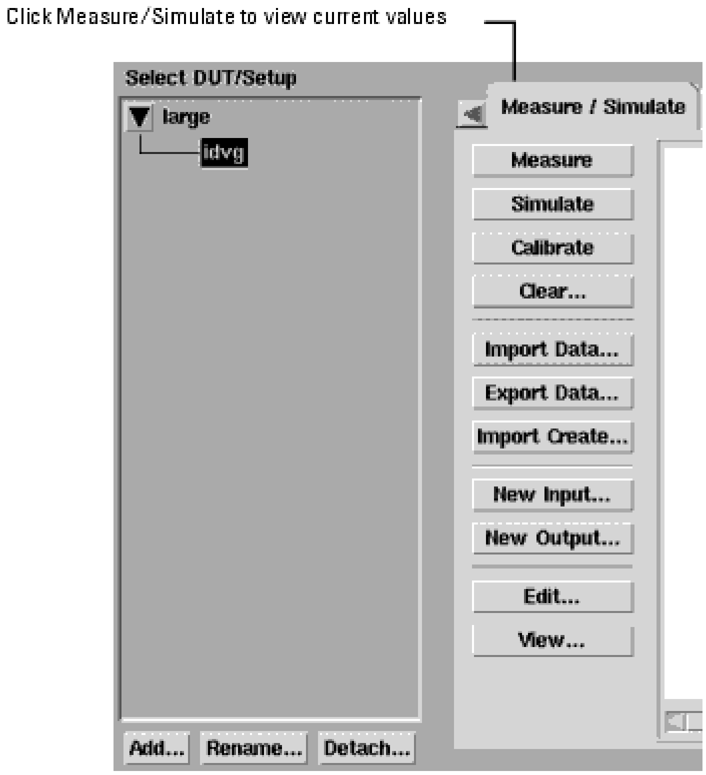


3. Click **OK** to add the output to the Measure/Simulate folder.

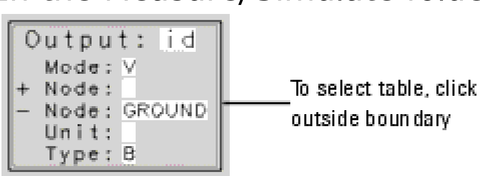


To edit inputs and outputs:

1. In the DUTs-Setups folder, select the setup. Then select **Measure/Simulate**.

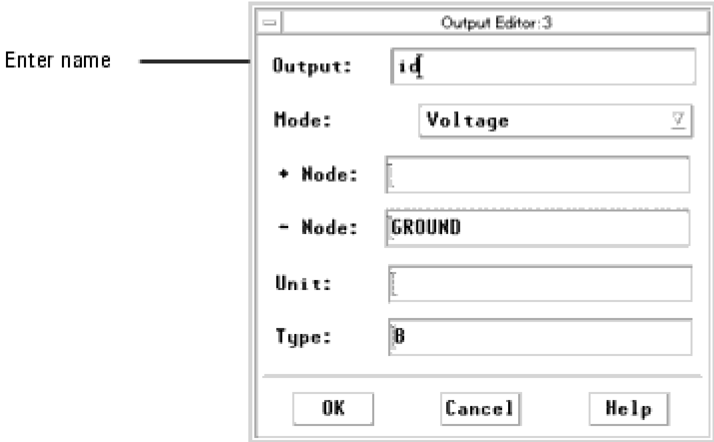


2. In the Measure/Simulate folder, select the Input or Output you want to modify.



3. Click **Edit** to open the Input or Output Editor.

4. Select the field you want to modify and edit the existing data.



5. Click **OK**.

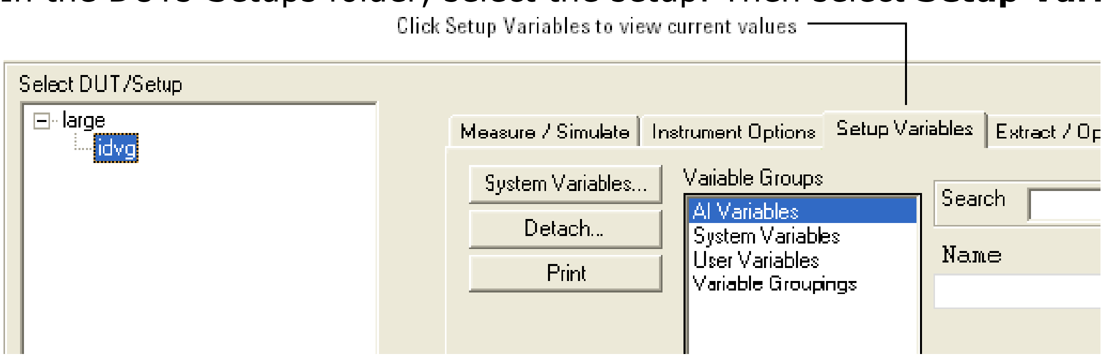
> **Note**
> Alternatively, you can edit directly in the table by selecting the field and typing the new values.

Defining Setup Variables

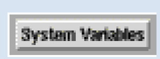
You can define any variables that are globally available to the setup.

To add or modify setup variables:

1. In the DUTs-Setups folder, select the setup. Then select **Setup Variables**.



2. Enter the names and values for all setup variables:
   - Click the first empty row in the Name field and type a name for the variable.
   - Press Tab to move to the Value field. Type a value for the variable.
   - Press Enter to accept the value and open another row.

> **Note**
> To view system defaults from the Setup Variables folder, select the  button.

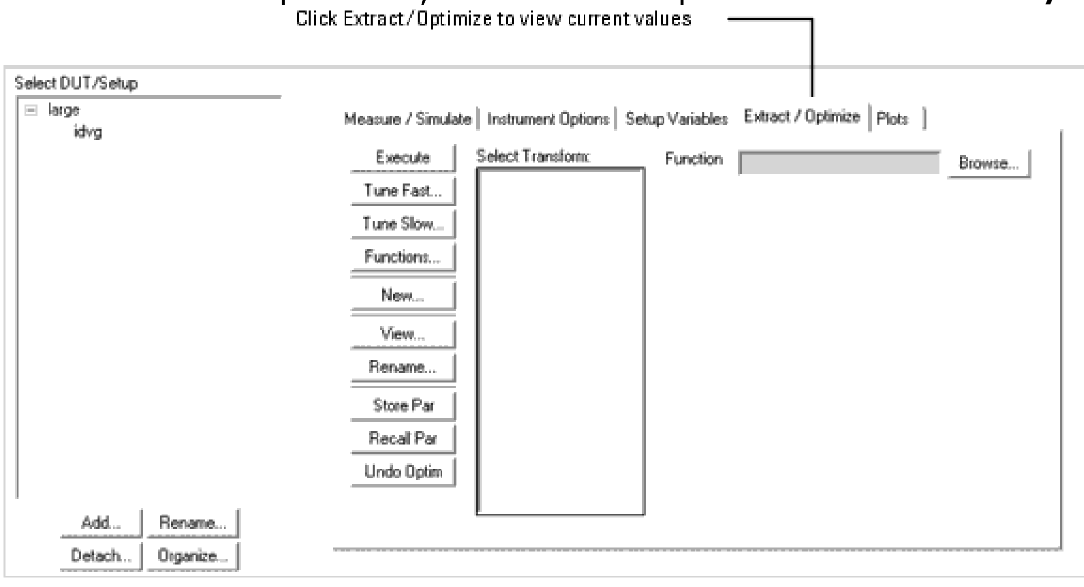
> **Note**
> You can copy setup variables from an existing model. Open the model and select the setup you want to copy. Select the **Setup Variables** folder. Select **Edit > Copy Setup/Variable Table**. In the selection dialog box, choose **Selected Variable Table**. In the new Model window, select the **Setup Variables** folder. Select **Edit > Paste Variable Table**.

Defining Transform Options

You can define transform options by adding new functions or by modifying existing functions.

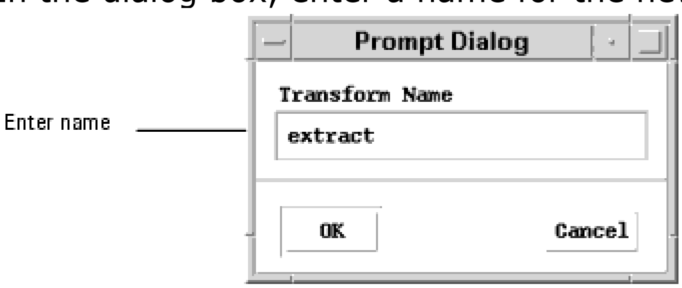
To add functions to a new setup:

1. In the DUTs-Setups folder, select the setup. Then select **Extract/Optimize**.

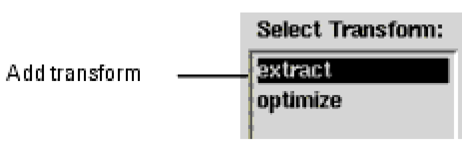


2. Click **New...**

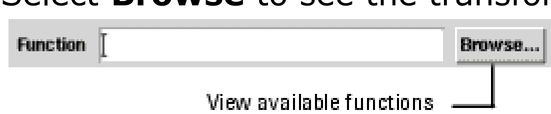
3. In the dialog box, enter a name for the new transform in the Transform Name field.



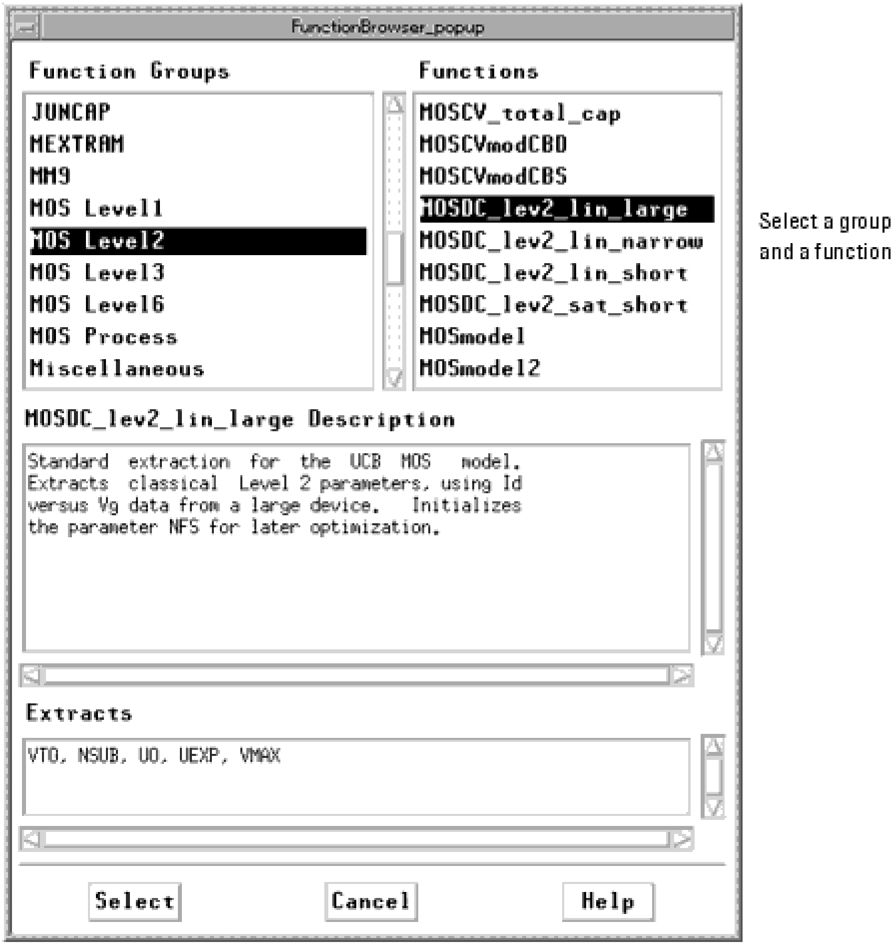
4. Click **OK** to add the transform to the list.



5. Select **Browse** to see the transforms provided with the program.



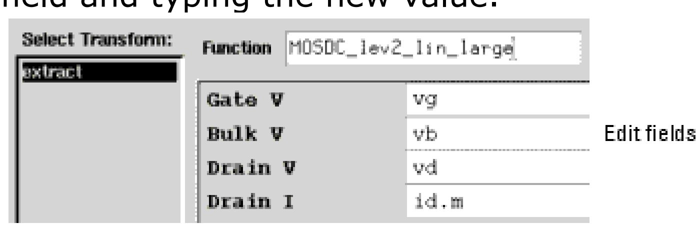
6. Choose a Function Group and a Function from the lists.



7. Choose **Select** to assign the selected function to the transform.

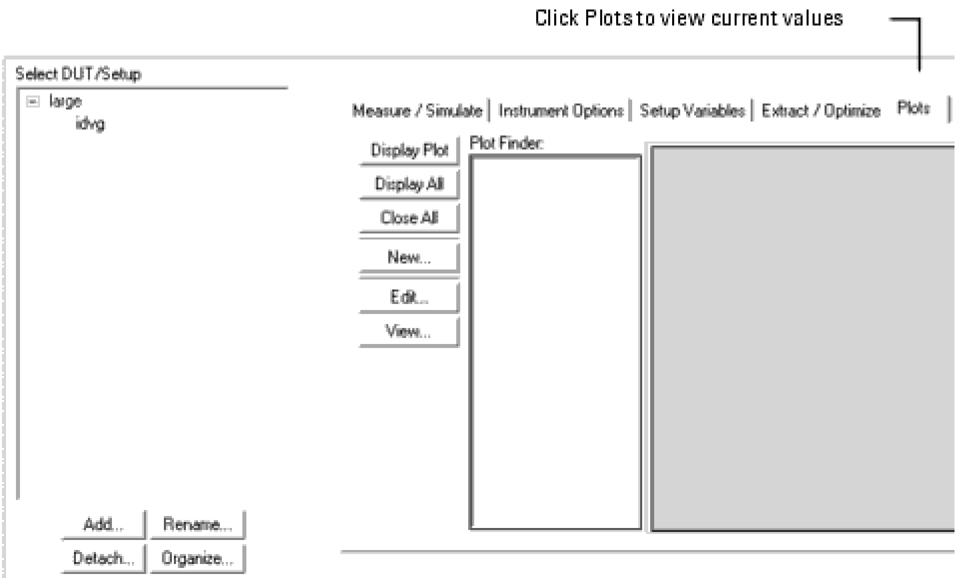
To edit functions:

1. In the DUTs-Setups folder, select the setup. Then select **Extract/Optimize**.
2. Select the function you want to modify. Edit the values in the table by selecting the field and typing the new value.



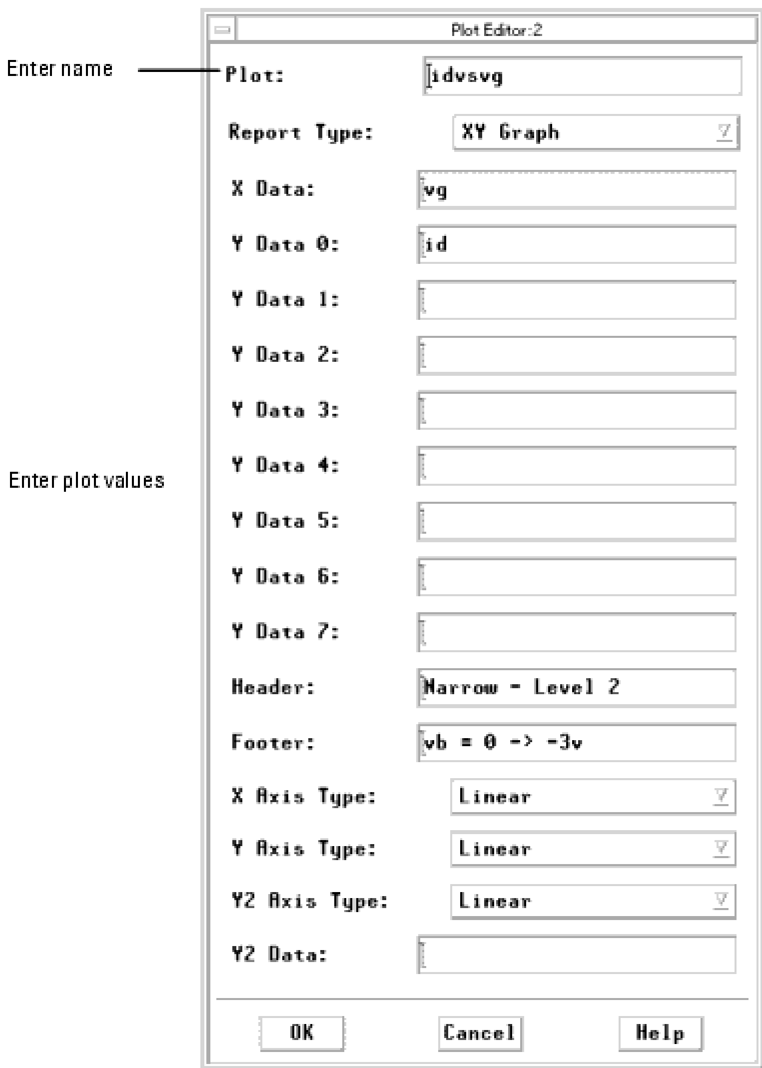
To add results to a new setup:

1. In the DUTs-Setups folder, select the setup. Then select **Plots**.

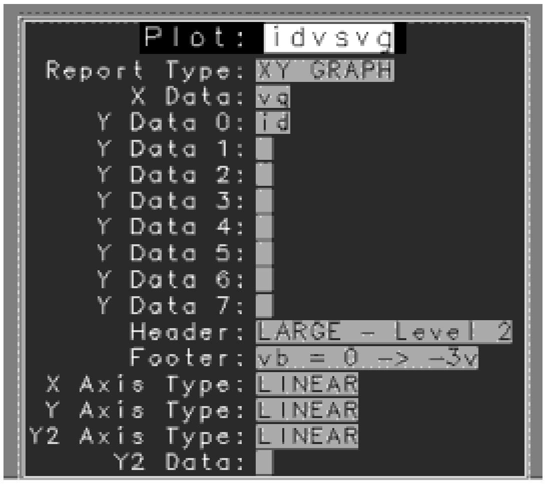


2. Click **New** to open the Plots Editor.

3. In the Plot field, enter a name for the new plot and complete the values as required.



4. Click **OK** to add the plot to the setup.



To edit plots:

1. In the DUTs-Setups folder, select the setup. Then select **Plots**.
2. Select the plot you want to modify.
3. Click **Edit** to open the Plots Editor.

4. Edit the values you want to change and click **OK**.

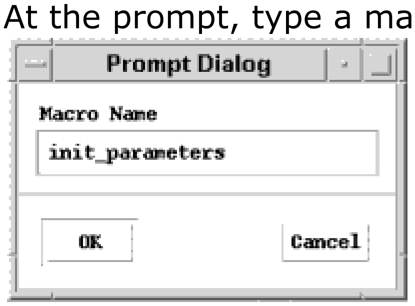
> **Note**
> Alternatively, you can edit directly in the table by selecting the field and typing the new values.

Adding a Macro to the Model

After defining a model, you can define several macros to help organize the characterization procedure. Macros allow any of the available IC-CAP menu operations to be combined and executed as a single operation. IC-CAP features a powerful macro language which is described in detail in *Creating and Running Macros* (prog).
To define a macro:

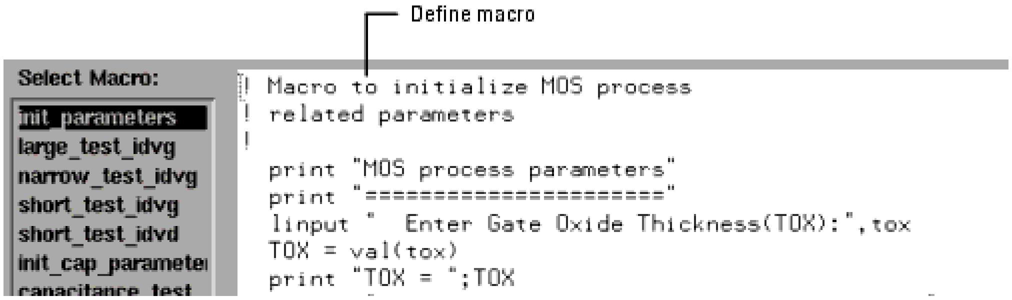
1. In the Model window, select **Macros**.
2. Click **New**.

3. At the prompt, type a macro name and choose **OK**.



The new name appears in the macro list.

4. To define the macro, type the program directly into the window.



Creating a New Model

The example in this section describes the procedure for defining a portion of the MOSFET model. The MOSFET model supplied with IC-CAP contains a variety of DUTs and Setups for characterizing the DC and Capacitance behavior of the device over a range of geometries. The model is described in *UCB MOS Level 2 and 3 Characterization* (mosfet). Several alterations have been made to the supplied model in order to illustrate advanced features, such as user-defined variables and test circuitry.

> **Note**
> These instructions assume that the necessary source and measuring instruments are already configured to the IC-CAP system. If this is not the case, refer to *Customization and Configuration* (custom).

Add a New Model to the Work Area

1. In the IC-CAP/Main window, select **File > New**. An icon representing a new untitled model appears in the IC-CAP/Main window.
2. Double-click the model icon to open the Model window.

Save the Model File

1. Select **File > Save As**.
2. In the dialog box, enter the design name, **nmos2**, and choose **OK**.

Define Model Variables

1. In the Model window, select **Model Variables**.
2. Click the first empty row in the Name field and type the variable name, `Compliance`.
3. Press Tab to move to the Value field. Type the variable value, `20m`.

Specify the Circuit Definition

1. In the Model window, select **Circuit**.
2. Enter the circuit description. For the *nmos2* model, the circuit is:

```
.OPTIONS ucb
M1 1=D 2=G 3=S 4=B MOSMOD L=2u W=3u
+ AD=9p PD=12u AS=9p PS=12u
.MODEL MOSCAP NMOS
+ LEVEL = 2.000
```

```
+ UO = 600
+ VTO = 0
+ NFS = 0
+ TOX = 100n
+ NSUB = 1E15
+ UCRIT = 10.0K
+ UEXP = 0
+ VMAX = 0
+ RS = 0
+ RD = 0
+ XJ = 0
+ LD = 0
+ DELTA = 0
+ NEFF = 1.00
+ NSS = 0
+ CGSO = 0
+ CGDO = 0
+ CGBO = 0
+ CBD = 0
+ CBS = 0
+ CJ = 0
+ MJ = 0.5
+ CJSW = 0
+ MJSW = 0.33
+ IS = 1.0e-14
+ PB = 0.8
+ FC = 0.5
```

The new circuit definition is parsed automatically when you click outside the window.

View Model Parameters

When a new model is read, parameter values are set to the defaults defined in the *MODEL* definition of the Circuit. To view the model parameters, select the *Model Parameters* folder. For this example, no changes are made to the model parameters.

Add a DUT

The MOSFET model contains three DUTs for DC characterization and two DUTs for Capacitance characterization. The procedure creates the DC DUT *large.*

1. In the Model window, select **DUTs-Setups**.
2. Click **Add**.
3. In the dialog box, click **Add New DUT**.
4. In the Enter New Dut Name field, type the DUT name, `large`.
5. Click **OK**.

View DUT Variables

To view the model parameters, select the DUT name **large**, then select the **DUT Variables** folder. For this example, no changes are made to the DUT Variables.

Specify DUT Parameters

The DC characterization of the MOSFET requires measurements on three DUTs. Each of these DUTs have different specifications for *L* and *W.* For the example, specify geometry parameters for the large device.

1. Select **DUT Parameters**.
2. Select the **Param Name Value** field and edit the DUT parameters for the device being measured. The DUT parameters are:

```
L - 50.00u
W - 50.00u
AD - 150.0p
PD - 106.0u
AS - 150.0p
PS - 106.0u
```

Specify a Test Circuit

A test circuit is usually not used with a MOSFET, but this example defines one. You can use a test circuit if you have a large amount of probe resistance that you do not want included in the model parameters. In this case, you can define a test circuit that has an additional resistor connected between each of the internal device nodes and the external nodes that are connected to the sources.

The example test circuit definition contains the following information:

- Line 1 contains the mandatory term .SUBCKT, which is required if a circuit contains more than one device (in this case, a transistor and four resistors). Line 1 also includes the arbitrary name of the test circuit and the identities of the four device nodes. The four device nodes are numbered as in the Model Circuit Definition (1 = C, 2 = B, 3 = E, 4 = S); however, alternate names can be used. If you do not assign a name to each node number, the system assigns the node number as the node name.
- Line 2 identifies the device (Q1), the internal nodes of the transistor (11, 12, 13, and 14), the name of the model (npn), and the AREA parameter. The model name must be entered precisely as it appears in the model icon.
- Lines 3 through 6 specify the four resistors that represent the additional resistance at each node. Each resistor is connected from an external Test Circuit node to the corresponding internal device node.
- Line 7 indicates the end of the .SUBCKT definition.

To define a test circuit:

1. In the DUT window, select **Test Circuit**.
2. In the test circuit window, type the test circuit data:

```
.SUBCKT large 1=D 2=G 3=S 4=B
M1 11 12 13 14 nmos2 L=50u W=50u
RDX 11 1 5.0
RGX 12 2 5.0
RSX 13 3 5.0
RBX 14 4 5.0
.ENDS
```

When a Test Circuit is defined for a DUT, the parameters in the Test Circuit appear in the DUT parameters folder and are editable.

Add a Setup

1. In the Model window, select **DUTs-Setups**.
2. Click **Add**.
3. In the dialog box, click **Add New Setup to Dut large**.
4. In the Enter New Setup Name field, type the setup name, `idvg`.
5. Click **OK**.

Specify Setup Components

To perform the measurement and simulation, four input voltages and one output current are specified. In the case of the *idvg* setup for the *large* DUT of the MOSFET, the drain current, *id*, of the device, is monitored as a function of the gate voltage, *vg,* and the substrate (bulk) voltage, *vb.* The drain voltage, *vd,* and the source voltage, *vs,* are held constant.

For each of the components added to the setup, fill in a table of information that defines the desired behavior. This example presents only a small subset of the available capabilities possible in a setup. To edit a setup:

1. In the DUT-Setups window, select **large/idvg**.
2. Select **Setup Variables** and define any variables you want to apply to the setup.
3. Select **Measure/Simulate**.
4. Click **New Input** to open the Input Editor.
5. In the Input field, enter the new input name, `vg`.
6. Complete the value entries as shown below:

Mode	**Voltage**
+ Node	G
- Node	GROUND
Unit	SMU2
Compliance	10.00u
Sweep Type	Linear
Sweep Order	1
Start	0.000
Stop	5.000
# of Points	21
Step Size	250.0m

   The value in the *Compliance* field is a reference to the global compliance variable that was defined for the setup variables.

7. Click **OK** to add the input to the Measure/Simulate folder.
8. Add the input `vb`. Complete the value entries as shown below:

Mode	Voltage
+ Node	B
- Node	GROUND
Unit	SMU4
Compliance	1.000m
Sweep Type	Linear
Sweep Order	2
Start	0.000
Stop	-3.000
# of Points	3
Step Size	-1.500

The value in the *Compliance* field is a reference to the global compliance variable that was defined for the setup variables.

9. Add the inputs that remain constant, `vd` and `vs`, by completing the value entries as shown below:

	Input vd	Input vs
Mode	V	V
+ Node	D	S
- Node	GROUND	GROUND
Unit	SMU1	SMU3
Compliance	100.0m	100.0m
Sweep Type	CON	CON
Value	100.0m	0.000

When you change the *Sweep Type* from *LIN* to *CON* the fields at the bottom of the table used to specify a linear sweep are replaced by a single *Value* field. Similarly, a change in the *Mode* alters the structure of the upper portion of the table.

10. Add the output `id` by completing the value entries as shown below:

Mode	I
To Node	D
From Node	GROUND
Unit	SMU1
Type	B

When you change the *Mode* to *I,* the names of the node fields of the table change accordingly.

Add Transforms

When the measurement has been made, values for the model parameters that control the *large* DUT behavior are extracted, and, optionally, the parameter values are optimized to achieve the best possible fit between measured and simulated data.

To accomplish this, you add a transform that performs the extraction and a transform that performs the optimization.

1. In the setup window, select **Extract/Optimize**.

   In the Function field, click **Browse**. Select the **Function Group MOS Level2** and the Function **MOSDC_lev2_lin_large**.

   In the list of transforms, select **extract**. Complete the values as shown below:

Gate V	vg
Bulk V	vb
Drain V	vd
Drain I	id.m

2. In the list of transforms, select **optimize**.

   For Inputs, complete the values as shown below:

Target	id.m	
Simulated	id.s	
Weight	1.000	1.000

   For Parameters, complete the values as shown below

Name	Min	Max
VTO	0.000	10.00
NSUB	1.000T	1.000E+18
UO	100.0	1.000K
UEXP	0.000	1.000
	1.000f	1.000MEG

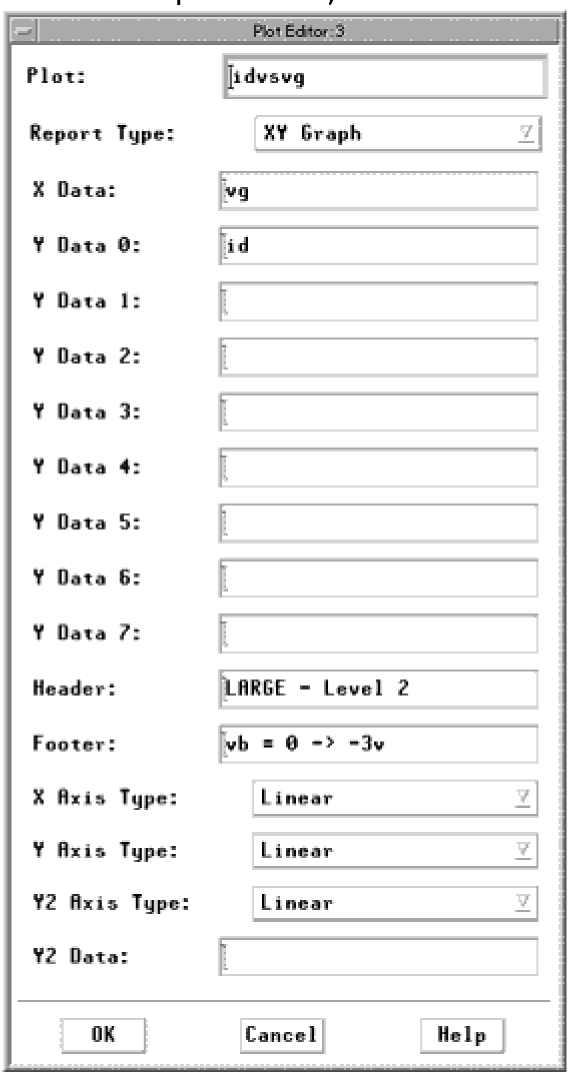
   For Options, complete the values as shown below:

Y Lower Bnd	0.000	Rank 1 Flag	Yes
Y Upper Bnd	0.000	Param Delta	1.000m
X Lower Bnd	0.000	Param Print	A
X Upper Bnd	0.000	Error Print	A
Comb Filter	1	Sens Print	1
RMS error	10.00m	Normal Sens	Yes
Max error	10.00m	Optim Mode	L
Abs Err Flag	Yes	Rand Std Dev	300.0m
Param Tol	1.000m	Rand Reward	500.0m
Function Tol	1.000m	Rand Penalty	500.0m
Max Evals	50	Rand Seed	0
Extract Flag	No	Rand Iters	25

The tables adjust depending on the functions selected.

Finally, in order to view the data and compare the measured and simulated results, add a plot to the setup.

1. In the setup window, select **Plots**. Define the *idvsvg* plot as shown below:

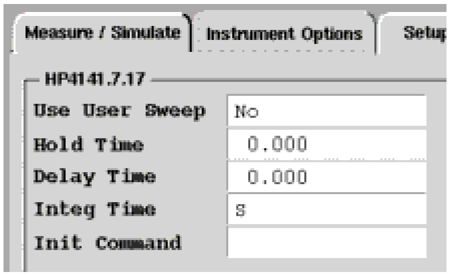


The plot table adjusts according to the value of the *Report Type* field.

2. Select **Instrument Options** to enter the instrument options values. Instruments used in each setup are determined from the instrument unit names specified in the Unit fields of the Input and Output tables.

Usually only one DC measurement instrument is used in this setup. When using the HP 4141, the window looks similar to the following figure.

Instrument Options for HP 4141



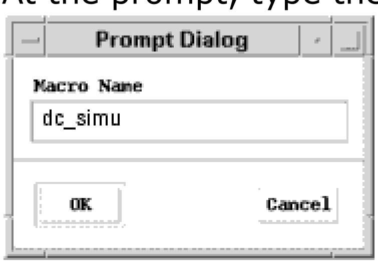
Add a Macro

You can add a macro to simplify simulations. The example for the MOSFET defines a macro that performs all of the DC simulations together. This macro calls the *Simulate* function for the *large, narrow,* and *short* DUTs.

To define a macro:

1. In the Model window, select **Macros**.
2. Click **New**.

3. At the prompt, type the macro name, `dc_simu`.



4. Click **OK**. The new name appears in the macro list.
5. To define the macro, type the program directly into the window.

```
menu_func("large", "Simulate")
menu_func("narrow", "Simulate")
menu_func("lshort", "Simulate")
```

To execute the macro, click **Execute**. 

When you execute this macro, all three DC DUTs (large, narrow, and short) are simulated.

> **Note**
> You can also execute macros from the Model window's Macro menu. Click on **Macros > Execute**, the click on the macro you want to execute. This is useful if you want to execute a macro, and you do not have the Macro folder open.

Create Other DUTs and Setups

You can create other DUTs and setups by repeating the example procedure. Or, you can build a complete model by copying portions of the supplied model and then modifying any parts of the new setup that needs to be changed to achieve the desired behavior. For the MOSFET model, you can copy the entire *large* DUT to two new DUTs named *narrow* and *short.* In the *short* DUT, the *idvg* setup can be copied to a new Setup named *idvd.* After making the necessary modifications, you have the complete set of DUTs and setups necessary for DC MOSFET characterization. For details, refer to Copying Parts of a Model.

DUT or Setup Editing

Models usually contain several DUTs. DUTs contain groups of Setups that have a similar physical connection to the device. If two Setups require different parameters or different test circuits, they must belong to different DUTs.

See also: *Defining DUT Variables* (intro).

Adding a DUT or a Setup

To add a DUT to a model:

1. With the model file open, click the **Add** button at the bottom of the DUT/Setup panel.
2. In the dialog box that appears, select **Add New DUT**.
3. Enter a name for the new DUT and click **OK**. The new DUT appears at the bottom of the DUT/Setup panel.

To add a Setup to a DUT:

1. With the model file open, and the appropriate DUT selected, click the **Add** button at the bottom of the DUT/Setup panel.
2. In the dialog box that appears, select **Add New Setup**.
3. Enter a name for the new Setup and click **OK**. The new Setup appears for that DUT. The new Setup appears below any existing setup(s).

See Organizing DUTs and Setups for an alternate way to add DUTs and Setups to a model.

Deleting a DUT or a Setup

To delete a DUT:

1. Select the DUT you want to delete.
2. Choose **Edit > Cut** and a dialog box appears.
3. Select the DUT option and click **OK**.

To delete a Setup:

1. Select the setup you want to delete.
2. Choose **Edit > Cut** and a dialog box appears.
3. Select the Setup option and click **OK**.

See Organizing DUTs and Setups for an alternate way to delete DUTs or Setups from a model.

Copying/Pasting a DUT or Setup

To copy and paste a DUT:

1. Select the DUT you want to copy.
2. Choose **Edit > Copy** and a dialog box appears.
3. Select the DUT option and click **OK**.
4. Choose **Edit > Paste** and the copied DUT is added after any existing DUTs.
5. Rename the DUT/Setups as needed (see Renaming a DUT or Setup).

To copy and paste a setup:

1. Select the setup you want to copy.
2. Choose **Edit > Copy** and a dialog box appears.
3. Select the Setup option and click **OK**.
4. Select the DUT you want to copy the setup to and choose **Edit > Paste** Setup. The copied setup is added below any existing setups of that DUT.
5. Rename the setup as needed (see Renaming a DUT or Setup).

Renaming a DUT or Setup

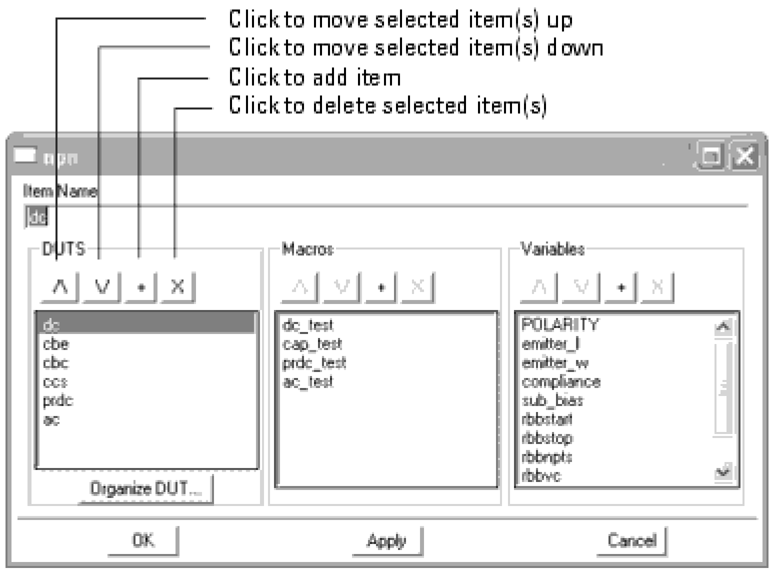
When you paste a DUT or Setup, the copy retains the name of the original, plus an underscore and a number (automatically incremented).
To rename a DUT or Setup:

1. Select the DUT or setup you want to rename.
2. Click the **Rename** button and a dialog box appears.
3. Type a new name for the DUT or setup and click **OK**. The name is updated immediately.

See also: Organizing DUTs and Setups

Organizing DUTs and Setups

Selecting **Tools > Organize Model** from a Model window displays the following dialog box:



This dialog box enables you to move, add, delete, or rename DUTs, macros, and variables.

To move an item:

1. Select one or more items:
2. Above the selected items, click the move up icon to move the items up in the list, or click the move down icon to move the items down in the list. Each click moves the selected items one position.

To add an item:

1. Choose where you want to add the item in the list:
    - Select an item if you want to add an item after that position.
    - Do not select an item if you want to add an item to the first position.

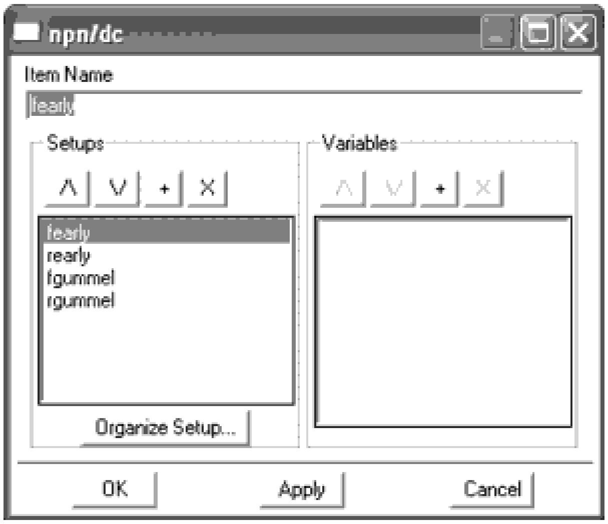
- Click the plus icon.
- A new item named Untitled is added. Type the desired name in the Item Name field then select the Enter key.

To delete an item:

1. Select one or more items:
2. Click the delete icon.

To display a dialog box that enables you to move, add, delete, or rename a DUT's setups and variables:

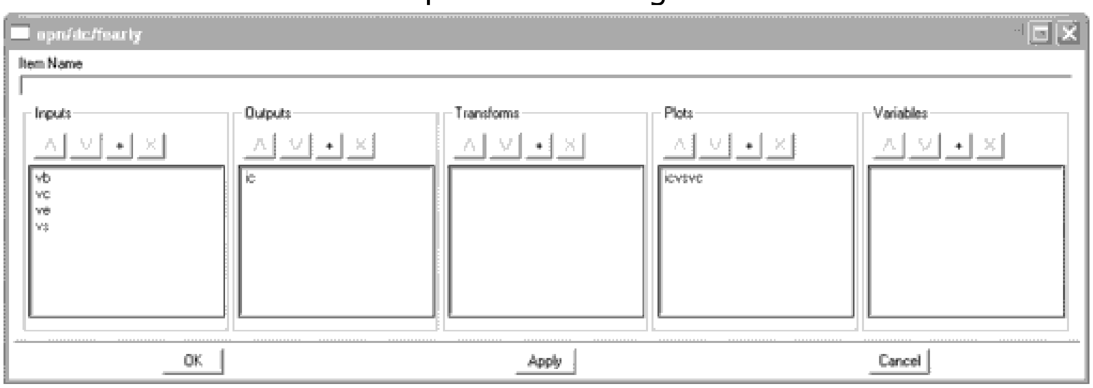
1. Select a DUT.
2. Click the **Organize DUT** button. This displays the following dialog box, which functions in the same manner as the previous dialog box.



Note
You can also display this dialog box by selecting a DUT in the model file, then clicking the **Organize** button at the bottom of the DUT/Setup panel.

To display a dialog box that enables you to move, add, delete, or rename a Setup's inputs, outputs, transforms, plots, and variables:

1. Select a Setup.
2. Click the **Organize Setup** button. This displays the following dialog box, which functions the same as the previous dialog box.



Note
You can also display this dialog box by selecting a Setup in the model file, then clicking the **Organize** button at the bottom of the DUT/Setup panel.

To apply you changes and leave the dialog box open, select the **Apply** button.

To apply your changes and close the dialog box, select the **OK** button.
To close the dialog box without making changes, click the **Cancel** button.

Defining a Test Circuit for a DUT

To define a test circuit:

1. Select the DUT for which you want to define a test circuit.

2. Create the definition by typing in the text area or by importing an existing text file.
3. When the file is complete, choose **Parse**.

> **Note**
> When you import text, the file is automatically parsed.

Defining DUT Parameters

To define parameters for a specific DUT:

1. Select the DUT and click to open the DUT Parameters folder.
2. Click once to activate the text field of a parameter you want to change.
3. Position the cursor as necessary, click again and use the backspace key to erase existing values and retype, or double click to highlight and retype the entire entry.

Tips:

- Use Detach to keep the parameters visible and editable, while viewing other parts of the model. When you are done, close the detached window and any changes made there are reflected in the Model window.
- Use Memory Store All to temporarily store the parameter set prior to extraction; use Memory Recall All to retrieve them if the extracted values are unacceptable.
- Use Reset All to overwrite DUT parameter values with circuit parameter values.
- Use Update Circuit to overwrite circuit parameter values with DUT values.

Setup Editing

Setups contain information for performing specific measurements and simulations on a DUT. The individual components of a Setup are:

- Inputs
- Outputs
- Extraction/Optimization Specifications (Transforms)
- Plots
- Variables
- Instrument Options

See also: Assigning Values to Setup Variables and Organizing DUTs and Setups

Input/Output Editing

The inputs and outputs of a Setup define the various currents, voltages, etc. you want to monitor with respect to the associated DUT.
To add a new input or output:

1. Click **New Input** or **New Output**.
2. Fill in all fields as necessary and click **OK**.

To delete an input or output:

1. Select the input or output.
2. Right click and select **Cut**. The selected input/output is deleted.

To edit an input or output:

1. Double-click the input or output or select it and click the **Edit** button in the Measure/Simulate folder or use the on-screen editor.
2. Make all the necessary changes in the dialog box and click **OK**.

   Alternatively, you can edit an input or output directly using the on-screen editor.

See also: Organizing DUTs and Setups

Variables

Several variable names are reserved by IC-CAP and cannot be assigned as user-defined variables. You can assign values to reserved variables and define your own variables at several different levels (variables at lower levels inherit their values from variables above them):

- Global
- DUT level
- Model level
- Setup level

You define global variables through the IC-CAP Main window. These variables apply to all Models, DUTs, and Setups unless you explicitly set the variables differently at the Model, DUT, or Setup level.
You define Model, DUT, and Setup variables through the Model window:

- Model variables apply to all DUTs and Setups of that Model unless you explicitly set the variables differently for individual DUTs and/or Setups.
- DUT variables apply to all Setups of that DUT unless you explicitly set the variables differently for individual Setups.
- Setup variables apply only to that Setup.

Assigning Values

Assigning Values to Global Variables

To assign values to global variables:

In the Main window, choose **Tools > System Variables** and the Variables window appears.

To create user-defined variables:

Type the names and values in the fields provided. Additional fields appear as you specify values.

To assign values to supplied variables:

- Type their names and values in the System Variables window or click **System Variables** and a dialog box appears.
- Select the appropriate category and the variables in that category are displayed.
- Click to select a variable and a description of that variable is displayed.
- Type the appropriate value in the Value field and click **Apply**. The variable name and its value appear in the Variables window.
- Continue assigning variable values as needed. When you are finished, click **OK**.

Assigning Values to Model Variables

To define variables at the model level:

Click to open the Model Variables folder.

To create user-defined variables:

Type the names and values in the fields provided. Additional fields appear as you specify values.

To assign values to supplied variables:

- Type their names and values in the System Variables window or click **System Variables** and a dialog box appears.
- Select the desired Variable Type and a list of variables of that type appears.
- Select the variable. A description is displayed.
- Enter the desired value for that variable and click **Apply**.
- Continue defining variables in this manner, clicking **Apply** to effect each change. When you are through defining variables, click **OK**.

Assigning Values to DUT Variables

To define variables for a DUT:

Click to open the DUT Variables folder.

To create user-defined variables:

Type the names and values in the fields provided. Additional fields appear as you specify values.

To assign values to supplied variables:

- Type their names and values in the System Variables window or click **System Variables** and a dialog box appears.

- Select the desired Variable Type and a list of variables of that type appears.
- Select the variable. A description is displayed.
- Enter the desired value for that variable and click **Apply**.
- Continue defining variables in this manner, clicking **Apply** to effect each change. When you are through defining variables, click **OK**.

Assigning Values to Setup Variables

To define Setup variables:

Click to open the Setup Variables folder.

To create user-defined variables:

Type the names and values in the fields provided. Additional fields appear as you specify values.

To assign values to supplied variables:

- Type their names and values in the System Variables window or click **System Variables** and a dialog box appears.
- Select the desired Variable Type and a list of variables of that type appears.
- Select the variable. A description is displayed.
- Enter the desired value for that variable and click **Apply**.
- Continue defining variables in this manner, clicking **Apply** to effect each change. When you are through defining variables, click **OK**.

See also: Organizing DUTs and Setups

Creating User-defined Variables

To create user-defined variables:

1. Select **Tools > System Variables** in the Main window if you are creating global variables, otherwise click to open the Model, DUT, or Setup Variables folder, as desired.
2. In the System Variables window that appears, type the names and values in the fields provided. Additional fields appear as you specify values.

Defining Instrument Options

The Instrument Options table is available once you specify the unit names (from the Hardware Setup configuration dialog) in the Inputs and Outputs, as needed. Once you define your instrument options, you can save them to file and read in this file anytime as needed.
To save instrument options:

1. With the Instrument Options folder active, choose **File > Save As**.
2. In the dialog box that appears, select **Instrument Options** (.iot).

3. If desired, choose another directory by typing or using the Browser.
4. Provide a filename (the extension is appended automatically) and click **OK**.

To read a file containing previously saved instrument options:

1. With the Instrument Options folder active, choose **File > Open**.
2. In the dialog box that appears, select **Instrument Options (.iot)**.
3. If desired, choose another directory by typing or using the Browser.
4. Provide a filename (the extension is appended automatically) and click **OK**.

Circuit Modeling

Circuit modeling is a natural extension of single device modeling. With IC-CAP's flexible structure, it is as easy to measure and characterize a multicomponent circuit as a single device. This section provides details for performing circuit modeling; typical applications are also provided to use as a guide for meeting specific circuit modeling requirements.

Definition of an IC-CAP Circuit

IC-CAP defines a circuit as any connection of two or more components. Previous sections have dealt primarily with single devices such as bipolar, GaAs or MOS transistors. An IC-CAP circuit can be a simple two-resistor voltage divider or a complex operational amplifier or A/D converter.

The circuit, like a single device, is specified in the Circuit folder of the model window using SPICE compatible circuit definition syntax. All circuit decks in IC-CAP begin with the *.SUBCKT* subcircuit definition and end with the *.ENDS* statement. Circuit modeling allows more accurate solutions to many single device modeling requirements and expands the level of systems modeling possible.

IC-CAP Circuit Modeling Operations

With IC-CAP, every type of characterization operation performed on a single component can also be performed on a circuit. IC-CAP allows easy measurement of circuit characteristics, extraction and optimization of model parameters, and simulation of the circuit's performance. Measurement and simulation operations use the same setup information as single components. Extraction and optimization operations enable more options for methods of obtaining model parameters. These operations can be performed on the circuit as a whole or on any sub-component of the circuit. This is explained in the section, Circuit Parameter Extraction.

Defining a Circuit

The process of defining a circuit in IC-CAP is similar to defining a single device. The main difference is the interconnection of the components and the use of subcircuit lines to define the circuit block. For detailed information on defining circuits, refer to *SPICE Simulators* (sim), *SPECTRE Simulator* (sim), or *Saber Simulator* (sim)).

Supported Circuit Components

Circuits in IC-CAP support the standard components that can be simulated with SPICE:

Passive elements R, L, C, Transmission lines
Semiconductors Bipolar, MOS, GaAs, JFET, Diode
Sources V, I, VCVS, VCIS, ICVS, ICIS

The syntax for defining a circuit in IC-CAP is similar to a SPICE simulation input deck.

Each line contains a component, its node numbers, value, and (if applicable) an associated model name reference. Proper specification and use of these components is critical to the success of circuit simulation and parameter extraction.

In general, independent voltage sources are specified as inputs within a given setup. This allows you to specify their values and use them in additional numerical or graphic analysis. Some of the differences between SPICE and IC-CAP circuit definitions are listed.

- The .OPTIONS statement (if used) must be the first line in the circuit description. All options must be on one line (no continuation).
- The next line of the circuit is .SUBCKT.
- A TITLE specification is automatically generated by IC-CAP and should not be included in the circuit definition.
- The last line of the circuit is .ENDS
- An .END statement is automatically generated by IC-CAP and should not be included in the circuit definition.

The following figure shows an example circuit description. This circuit defines the input section of an ECL OR/NOR logic gate. (ECL OR/NOR Schematic Diagram shows the schematic.) This circuit is referenced several times in this section. You can create it using the circuit editor or read it from the file $ICCAP_ROOT/data/ECLornor.mdl.

Circuit Description for an ECL OR/NOR Logic Gate

```
.SUBCKT ECLORNOR  1=IN1 2=IN2 3=OR 4=NOR
+ 5=VCC 6=VEE 7=VREF
* ECL OR/NOR LOGIC GATE
Q1   4  1  8  NPN1
Q2   4  2  8  NPN1
Q0   3  7  8  NPN2
.MODEL NPN1 NPN
+ IS = 2E-14  NF = 0.998  BF = 120
+ RB = 225  CJC = 300p  TF = 20p
.MODEL NPN2 NPN
+ IS = 4E-14  NF = 0.998  BF = 120
+ RB = 110  CJC = 530p  TF = 18p
RL1  5  4  300
RL0  5  3  300
RIEE 8  6  1.2K
.ENDS
```

When you enter the circuit description in the Circuit folder of the model window, moving the mouse out of the Circuit folder automatically causes the circuit to be parsed, that is, the specified circuit elements are read and entries are created for them in Model Parameters. When they are added initially, they assume the value specified in the circuit description. To change a value subsequently, you must change it in Model Parameters. To change all entries in Model Parameters to the values in the circuit description, choose **Reset**.

Note the difference in the Parameters table parameter names for a transistor in a circuit. In a single transistor circuit, the model parameter names of the transistor are the entries in the Parameters table. In a multi-component circuit the transistor's model parameters must be associated with a specific model, so the parameters take on a prefix of that model's name. Thus, the forward Beta model parameter *BF* for a model named *NPN1* is listed in the Parameter Editor as *NPN1.BF.* In the example above, transistors *Q1* and *Q2* both use the *NPN1* model, while transistor *Q0* uses the *NPN2* model.

Circuit Measurement

The process of measuring a circuit in IC-CAP is identical to measuring a single device. The circuit stimuli and responses are specified in the input and output tables, respectively, of the Measure/Simulate folder. You can perform a measurement by clicking Measure in the Measure/Simulate folder the DUT or Setup levels. In performing measurements on circuits, there are several additional items not found in single component measurement.

Multiple Instrument Names

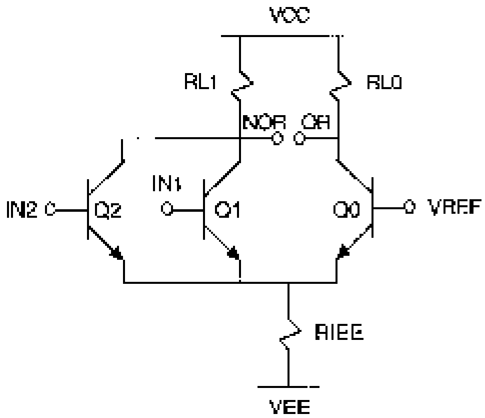
In measuring a single component, it is common to use only one DC source and measurement instrument because only four terminals are involved. The typical circuit can have more than four terminals and require several DC source and measurement instruments. Any number of instruments of the same or similar type can be connected to the circuit under test as long as they are entered in Hardware Setup. When using multiple instruments, each of their units must have a unique name.

Isolating Circuit Elements for Measurement and Extraction

The characterization of a circuit may require the measurement and modeling of several sub-circuit elements. The accuracy of the sub-circuit model generated is dependent upon how well that circuitry can be isolated from the rest of the overall circuit.

Examine the simplified schematic of the input to an ECL OR/NOR gate in the following figure. The input stage of this circuit is a differential amplifier with a collector resistor in each leg and a resistor for a current source. It is possible to characterize the individual transistors in this circuit by selectively biasing only the terminals that make it active and that keep other parts of the circuit in an off or latent state. In this case, biasing IN1, VCC, and VEE turns on the circuit that contains RL1, Q1, and RIEE. These components have now been isolated so that their model parameters can be extracted. This type of selective measurement allows characterization of individual or small groups of sub-circuit elements.

ECL OR/NOR Schematic Diagram



Circuit Parameter Extraction

Circuit parameter extraction is identical to single component parameter extraction through the use of Transforms. However, because circuits are custom in nature, most of the extraction routines must also be custom designed. With the availability of the Program function and optimize transforms, this is simple and quick to evaluate and execute. The critical factor in a successful circuit level parameter extraction is the ability to make a measurement and subsequent extraction involving only the dominant component parameters.

For a full model extraction of a single component, you will attain more accuracy if that device is available without any additional components connected to it. For most functional block level circuits however, a subset of the transistor model parameters is usually sufficient for studying circuit behavior.

Extracting Transistor Parameters Using Library Functions

In the explanation of a selective measurement on a sub-portion of a circuit in the previous section, Q1 and its neighboring resistors were isolated in the ECL logic gate. The forward active model parameters can be extracted from this measurement using the model extraction functions in the function list or by setting up a custom optimization. To access the functions, add transforms that use them to a setup that contains the measurement. It is possible to use the provided transistor extraction functions to obtain model parameters for devices connected into a larger circuit.

Because all models in a circuit have model parameters in the Parameters table with the model's name as a prefix, IC-CAP must be told which model to use with the extraction transforms. This is easily done by setting a variable in the model level variable table. Enter a variable in the table called EXTR_MODEL and set its value to the name of the transistor whose parameters are being extracted. When the extraction transforms are executed, IC-CAP refers to the correct Parameters table entry as it writes the extracted value back to the table. For example, to use a function list transform on the model *NPN1* mentioned above, add the following to the model level variable table:

```
EXTR_MODEL NPN1
```

Each time another transistor is used for an extraction, place its name in the value field. A more efficient method of extracting individual transistor models is to create an individual setup for each device. The variable table at the setup level can then include the EXTR_MODEL entry, keeping the transistor extraction local to that setup. This can also be done in an analogous way at the DUT level.

It is sometimes necessary to specify the particular DUT in a circuit that should be used in an extraction routine. For example, in a circuit that contains two MOSFETs there are two different sets of geometry parameters (L and W). For the extraction to work correctly, the EXTR_DUT variable must be set to the name of the transistor with the correct geometry parameters. Therefore, to characterize transistor *M1,* which uses model *NMOS1,* add the following to the model level variable table:

EXTR_DUT M1
EXTR_MODEL NMOS1

Situations where EXTR_DUT must be set can also arise when test circuits are defined. In this case, DUT parameters that normally appear without a prefix in the DUT Parameters table will include the transistor name from the Test Circuit as a prefix. For the extractions to use these parameters, EXTR_DUT must be set to the transistor name that is used in the test circuit.

Extracting Parameters Through Optimization

It is not always possible to adequately isolate a circuit component before using a standard extraction function. In these cases it is still possible to extract model parameters by using the optimize function. As with any extraction function, successful use of the optimizer requires that the parameters being optimized have a dominant effect over the simulation of the measured characteristics. Refer to *Optimizing* (sim), for more information regarding optimization.

In the OR/NOR gate shown in ECL OR/NOR Schematic Diagram, it is possible to use the optimizer to extract the values of NPN1.IS, NPN1.NF, NPN1.BF and RIEE. The following sequence of operations describes how to accomplish this.

1. Make an Ic and Ib versus V measurement between the NOR, IN1 and VEE terminals.
    1. Connect the VEE, VREF and IN2 terminals to constant voltage sources of 0V. This keeps the base-emitter diodes of Q2 and Q0 in an off state. Disconnect VCC from the circuit.
    2. Connect the NOR terminal to a voltage of approximately 1.0V.
    3. Sweep the voltage on the IN1 terminal so that the measured currents at the NOR (collector) terminal are in the 1nA to 1µA range.
2. Set up an optimization transform that optimizes the values of NPN1.IS, NPN1.NF, and NPN1.BF over the measured current. At low currents, RIEE has a minimal affect on the I-V relationship.
    - The target data is the measured Ic and Ib currents. The simulated data comes from the simulation of these currents.
    - The Parameters table contains NPN1.IS, NPN1.NF and NPN1.BF
3. Change the sweep voltage on the IN1 terminal so that the measured current at the NOR terminal is in the 10µA to 1mA range.

    - The measured current should deviate from an exponential function due to the debiasing effect from RIEE.
4. Set up an optimization transform that optimizes the value of RIEE over the measured current.
    - The target data is the measured Ic current. The simulated data comes from the simulation of this current.
    - The Parameters table contains only the circuit element RIEE.

After each of these measurements and optimizations has been executed, the Model Parameters table is updated with the extracted values of these elements.

Circuit Simulation

Circuit simulation is performed identically to single device simulation. The circuit usually has more inputs and outputs defined than a single device. In addition, the simulated circuit can use independent or controlled voltage and current sources that are defined within the Circuit Editor. When IC-CAP simulates a circuit, it first builds a complete SPICE deck from the circuit description and the setup table. The source names are built from the source type (V or I) and the nodes to which they are connected. Use of the simulation debugger can improve efficiency in performing successful simulations. Knowledge of how IC-CAP interacts with the SPICE simulators gives you more control over the options available for circuit simulation. For more information, refer to *SPICE Simulators* (sim).

One of the advantages of simulating circuits through IC-CAP is the increased levels of analysis available. IC-CAP allows a sweep of more parameters than with a stand-alone SPICE simulator. For example, it is possible to simulate a circuit's behavior over bias conditions, component values and temperature in the same simulation. Once a simulation is complete, you may further analyze the stimulus and response data with IC-CAP's numerical and graphic capabilities. The two-port simulation features enable you to study the high-frequency characteristics of a circuit using any of the S, H, Y, or Z 2-port parameters.

Design Optimization

Designing a circuit usually follows a path of defining a block level functional description, translating it into discrete circuit components, then optimizing those components for the required performance. This last stage can be simplified with the IC-CAP system. It is possible to specify the desired performance from a circuit and then let the IC-CAP optimizer find the best component and model parameters to satisfy it. The following is an overview of how to do this.

1. Enter a circuit with a first order estimate of the required parameters and component values.
2. Create a setup with the inputs and outputs that simulate the desired region of performance.
3. Simulate the circuit to create a set(s) of output data.
4. Copy the simulated data into the measured data set(s) in the outputs.
    1. Type the letter *S* in the *Type* field and press Return.
    2. Type the letter *B* in the same field to replace the letter *S.*

5. Save the desired outputs to files using the Write to File menu choice on each output.
6. Edit the files using any editor. Scan down the file to find the *type MEAS* data section. This is where the measured data begins. Edit the output values, replacing them with the desired performance values for the circuit. Save the file when done.
7. Read the file (and thus the new data) back into the outputs in IC-CAP using the *Read from file* command on the desired outputs.
8. Set up an optimization transform to find the required parameter and component values that best match the new measured data.

This type of design optimization can save many hours of iteration in fine-tuning high-performance circuit designs.

Test Circuits

When measuring a single device or a complex circuit you often require additional components for biasing, setting operating points, or tuning the high-frequency performance characteristics. Even when no additional components are required, there may still be some parasitic elements introduced by the connections of the device to the instrumentation. Examples of this are DC resistance in probe-to-wafer contacts, inductances in IC package bond wires, and the shunt resistances in the bias ports of network analyzers. The Test Circuit in IC-CAP allows you to include these elements when performing a simulation or optimization without including them in the main circuit description or device model.

Syntax

A circuit editor is available for each DUT. To access it, select Edit in the DUT Circuit folder. This produces a window that has both the DUT Parameters table and the test circuit editor. The mechanics of using this editor are the same as using the circuit editor.

The test circuit adds another level of circuit hierarchy to the overall system being measured and modeled. It is implemented through a circuit description that uses the SPICE subcircuit syntax. The example test circuit shown in the following figure adds a capacitor and resistor to the outputs of the ECL logic gate described earlier.

Test Circuit for an ECL Logic Gate

```
* Resistive / Capacitive circuits to
* simulate the effect of gate loading
.SUBCKT gateload 1=IN1  2=IN2  3=OR
+ 4=NOR  5=VCC  6=VEE  7=VREF
Xornor 1 2 3 4 5 6 7 ECLornor
Cor  3 0 1p
Ror  3 0 10MEG
Cnor 4 0 1p
Rnor 4 0 10MEG
.ENDS
```

This test circuit is added to the SPICE circuit deck each time a simulation is called or when an optimization that uses a SPICE simulation is performed. It does not modify the original model description in any way. The values of the elements in the test circuit can be modified in the DUT Model Parameters.

Hierarchical Modeling

The previous test circuit section on illustrated one way to include hierarchy in an overall circuit description. The test circuit, however, is at the highest level of hierarchy in an IC-CAP circuit. It is also possible to build a complete circuit by combining smaller circuit or transistor models into one subcircuit definition. This way, you can update the models of smaller subcircuits or individual components and have these changes automatically propagate into all circuits that use it.

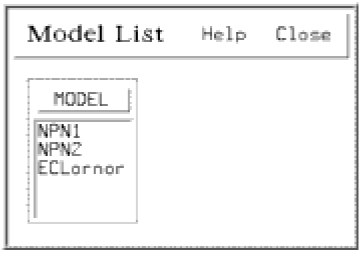
Circuits Built from Sub-models

The ECL logic gate defined in ECL OR/NOR Schematic Diagram uses two sizes of NPN transistors. Each transistor has a separate .MODEL card associated with it. These transistor model definitions can be removed from the logic gate circuit and reference other models currently active in IC-CAP. When a simulation is performed, IC-CAP includes these device models in the circuit definition.

To use external models, the models that you want to include must be in the IC-CAP model list. In the circuit definition that uses these model references, remove the .MODEL card. The model name used for the transistors should then be the same as the names in the IC-CAP model list. To use this technique for the ECL logic gate, read in models for the *NPN1* and *NPN2* transistors into IC-CAP. Then delete the two model cards in the logic gate circuit. The resulting model list and circuit description are shown in the following 2 figures.

This approach provides flexibility. It allows you to keep a standard library of device models to include in larger circuits requiring accurate device models. This allows you to quickly *cut-and-paste* different component models into circuits and compare performance. It also greatly reduces the size of the circuit definition that needs to be maintained.

Model List Window for Hierarchical Circuit Definition



Hierarchical Circuit Description for ECL OR/NOR Logic Gate

```
.SUBCKT ECLORNOR  1=IN1 2=IN2 3=OR 4=NOR
+ 5=VCC 6=VEE 7=VREF
* ECL OR/NOR LOGIC GATE
Q1   4  1  8  NPN1
Q2   4  2  8  NPN1
Q0   3  7  8  NPN2
```

```
RL1  5  4  300
RL0  5  3  300
RIEE 8  6  1.2K
.ENDS
```

Functional Circuit Blocks

Previous sections in this topic provided different aspects of the use of IC-CAP for modeling complete functional circuits. This section provides detailed examples of circuit models and custom model extractions you can create. These examples are provided to stimulate ideas for using IC-CAP to meet specific circuit modeling requirements.

Types of Circuits in IC-CAP

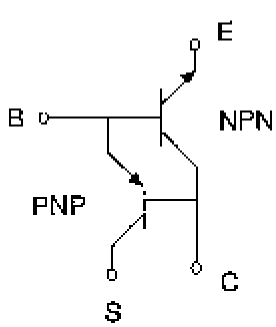
The types of circuits for measurement and simulation with IC-CAP are unlimited. Anything that can be simulated on a stand-alone SPICE simulator can be simulated with IC-CAP. In fact, any type of system that can be measured with the IC-CAP library of instruments can be characterized.

With the variety of components supported by SPICE, IC-CAP can be used to both characterize and design circuit modules. Classical examples of both single device and functional circuit modeling problems that are easily solved with IC-CAP are included.

Modeling the Reverse Active Region of an NPN Transistor

One of the constant sources of error in modeling the performance of a reverse active NPN transistor is the parasitic PNP formed by the base, collector, and substrate of the integrated structure. (This was briefly described in *PNP Transistors* (bipolar)) A simple solution to modeling this region of operation is to use the complete functional circuit displayed in the following figure.

NPN Transistor with Parasitic PNP



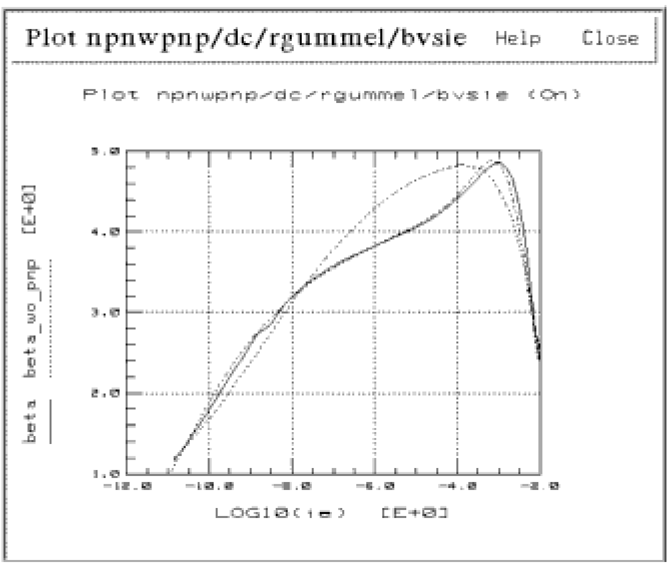
Model file *npnwpnp.mdl* is included as an example of solving this problem. It has a single DUT/Setup that measure and model the reverse active operation of an NPN transistor.

The previous figure shows that the emitter of the PNP steals current from the base terminal of the NPN. The single dominant parameter that models this PNP current flow is the saturation current IS. (Modeling the transistor at this point assumes that the DC NPN parameters have already been obtained using another model file. The *bjt_npn.mdl* model file has a complete set of DUTs and setups to perform this. For more information, refer to *Bipolar Transistor Characterization* (bipolar).)

The *rgummel* setup then uses an optimize function to simultaneously extract the reverse active NPN parameters BR, IKR, ISC, and NC and the PNP parameter IS. The optimization proceeds by simulating the compound device, which is represented as a 2-transistor circuit, and numerically adjusting these model parameters.

The plot in the following figure is of reverse beta versus emitter current. It includes the simulation of the single transistor reverse model and measurement and simulation of the 2-transistor compound structure. The result of this extraction is a near perfect agreement between measured and simulated data. Also examine the resulting magnitudes of both IS (large enough to not be negligible) and BR (much higher than for a single device extraction). This example shows the improvement you can attain in using a full circuit description to model an integrated device structure.

Reverse Beta versus Ie for Single NPN and Compound NPN-PNP Structure



Modeling an Operational Amplifier

The operational amplifier is included with IC-CAP as an example of how to do relatively complex macro modeling. It illustrates the simplification of a complex circuit to the necessary and sufficient components that enable it to be accurately represented. This model includes a Program transform that extracts model parameters from data sheet specifications of its performance. The inputs to this transform could also be measurements of the opamp's electrical characteristics. This same Program transform has also been converted into a standard IC-CAP function by writing it in the *C* programming language (in the *userc.c* module) and compiling and linking it into the system. The circuit chosen follows the model developed by Boyle, Cohn, Pederson, and Solomon [1]. This circuit model is in the *opamp.mdl* example file.

Opamp Macro Model

The stages in this opamp model are: non-linear differential input, intermediate linear gain, and output driver. These enable most of the possible operating regions of the complete circuit to be adequately simulated.

The input stage contains transistors Q1 and Q2 connected as a differential amplifier, biased with a fixed current source (see figure above). Q1 and Q2 provide both differential mode (DM) and common mode (CM) characteristics. Passive components in the input stage provide slew rate effects (C2, Ce, Re1, Re2), 0dB frequency control (Rc1, Rc2), DM excess phase (C1), and CM input resistance (R1).

The intermediate gain stage provides linear amplification through voltage controlled current sources Ga, and Gb, which model the differential gain of the opamp. Capacitor C2 controls the dominant pole. CM gain is modeled with the voltage controlled source that has the coefficient Gcm.

In the output stage, AC and DC output resistances are modeled with Ro1 and Ro2. Output drive voltage is supplied through diodes D1 and D2 and voltage controlled voltage source (Gc*v6*Rc). The independent voltage sources in series with diodes D3 and D4 clamp the opamp under conditions that would force the output voltage to one of the supply rails.

The development of this opamp model uses the concepts of simplification and buildup to reduce the number of active and passive components. For example, to maintain non-linear effects the input stage has been simplified to two transistors, and an independent current source has been substituted for the usual bias circuitry. In the interstage amplifier, *buildup* is used to emulate a circuit characteristic through alternate circuitry. The stage is modeled by two voltage controlled current sources and a capacitor for compensation. These techniques take a piece-by-piece approach to the development of a model. They can be applied to virtually any circuit or subcell.

Opamp Circuit Model

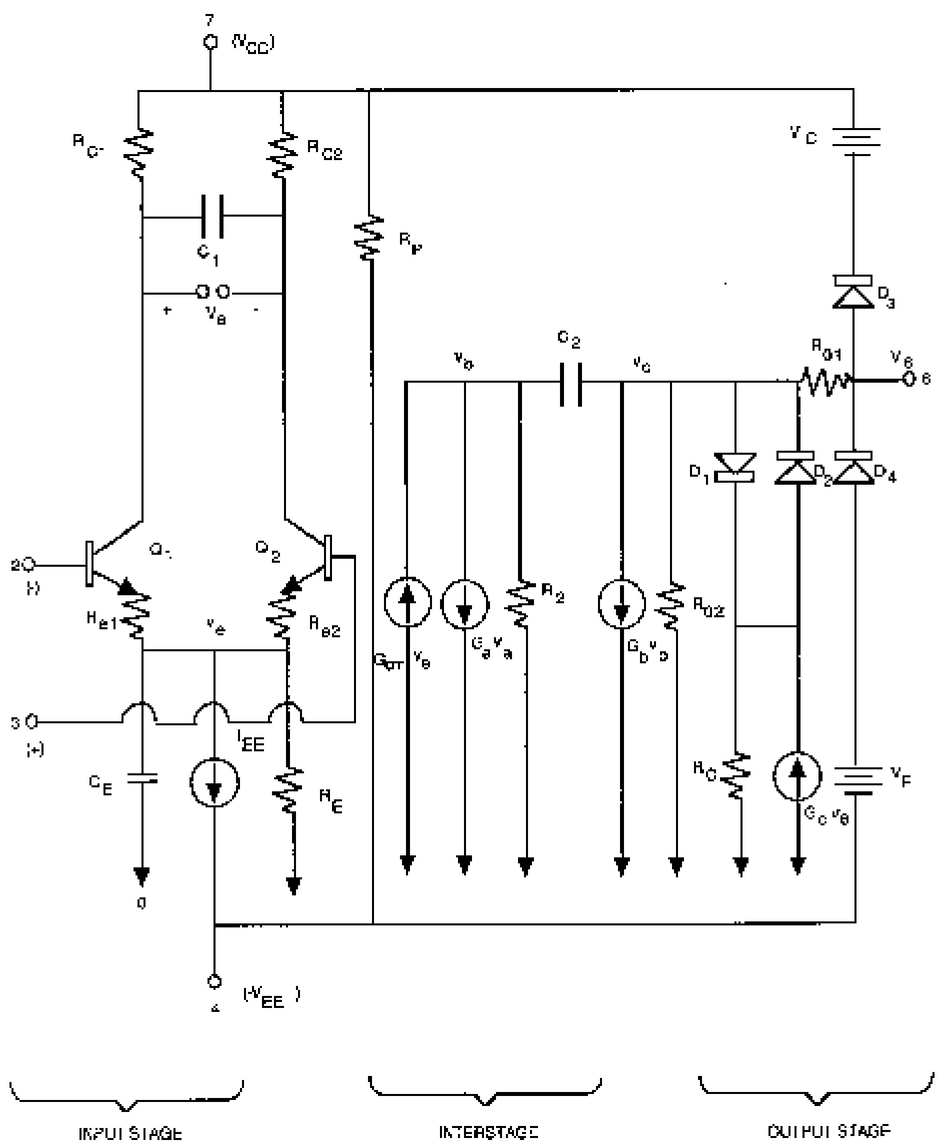
The following figure shows the macro model that represents the full opamp circuit, followed by circuit elements in order from the input to the output stage. Opamp Circuit Diagram shows the complete opamp model circuit definition used in this example.

Opamp Circuit Definition

```
.SUBCKT OPAMP_1 2 = VINP 3 = VINN
+ 4 = VEE 6 = VOUT 7 = VCC
Q1 10 2 12 NPN1
Q2 11 3 13 NPN2
.MODEL NPN1 NPN IS = 8.0E\-16 BF = 111.7
.MODEL NPN2 NPN IS = 8.31E\-16 BF = 143.6
RC1 7 10 5305
RC2 7 11 5305
C1 10 11 5.46p
RE1 12 14 2712
RE2 13 14 2712
RE 14 0 9.872m
CE 14 0 2.41p
RP 7 4 15.36K
```

```
GCM 0 15 14 0 6.28n
GA 15 0 10 11 188.6u
R2 15 0 123.4K
C2 15 16 30P
GB 16 0 15 0 247.5
RO2 16 0 42.87
D1 16 17 DMOD1
D2 17 16 DMOD1
.MODEL DMOD1 D IS = 8.0E\-16
RC 17 0 0.02129m
GC 0 17 6 0 46964.0
RO1 16 6 32.13
D3 6 18 DMOD2
D4 19 6 DMOD2
.MODEL DMOD2 D IS = 8.0E\-16
VC 7 18 1.803
VE 19 4 2.303
IEE 14 4 20.26u
.ENDS
```

Opamp Circuit Diagram



Inputs to the Opamp Macro Extraction

Inputs to the opamp model extraction describe its electrical performance. These characteristics can be measured on actual devices or obtained from data sheet specifications. Due to the flexibility of the origin of the inputs, you can experiment with the opamp's performance as it relates to the model elements that control it. The following table lists the inputs to the opamp extraction.

opamp Extraction Inputs

Input Name	Contents
Slew Rate +	positive-going slew rate
Slew Rate-	negative-going slew rate
Bias Current	average input base bias current
Bias Offset	input bias offset current
Volt Offset	input offset voltage
Av(DM)	open loop differential mode voltage gain
BW	unity gain bandwidth (Av(DM) * f(-3dB))
Excess Phase	excess phase at f(0dB) due to 2nd pole
CMRR (dB)	common mode rejection ratio
Rout	low-frequency output resistance
Rout-ac	high-frequency output resistance
Isc+	positive short circuit current
Isc-	negative short circuit current
Vout_max+	positive output voltage where opamp clamps Iout
Vout_min-	negative output voltage where opamp clamps Iout
Power Diss	quiescent state power dissipation
Vcc supply	positive supply voltage
Vee supply	negative supply voltage
Nom. Q.IS	nominal transistor and diode saturation current
R2	differential gain setting resistor
Comp. Cap.	compensation capacitance
Temp.(C)	temperature for macro extraction and input specification
Inputs PNP?	flag to switch the inputs to PNP transistors
Debug?	flag to turn on debug output during extraction

Extraction Equations for the Opamp Macro Model

The inputs to the macro model extraction described are used by the set of equations shown in the following figure to produce the model parameters. These equations are programmed into the *userc.c* module exactly as shown.

Because of the simplicity of the equations, they can also be entered into a Program transform using the Parameter Extraction Language. This allows experimentation with model extraction techniques before coding the final extraction in C and linking it to IC-

CAP. This has been done with the opamp macro model to provide a typical example of writing custom model extractions.

C Coded Opamp Extraction Equations

```
if ( is_pnp )
/* for pnp, pos. and neg. slew-rates are interchanged in calculations */
     { tmp = sr_plus ; sr_plus = sr_neg ; sr_neg = tmp ; }
vt = 25.85E-3 * (temp/27);
isc = ( isc_p + isc_n ) / 2 ;
is1 = isd3 = isd4 = is ;
sr_plus * = 1.0E6; /* convert slew rates to Volts/sec */
sr_neg * = 1.0E6;
ic1 = ic2 = (c2/2) * sr_plus ;
ce = 2*ic1/sr_neg - c2 ;
ib2 = ib + ibos/2 ; /* ib2-ib1 = ib+ - ib- = ibos */
ib1 = ib - ibos/2 ;
if (is_pnp)
/* bias currents will have opposite signs with pnp input stage */
     { ib1 = -ib1 ; ib2 = -ib2 ; }
beta1 = ic1/ib1 ;
beta2 = ic2/ib2 ;
iee = (( beta1+1 )/beta1 + (beta2+1)/beta2 ) * ic1 ;
re = 200/iee ;
is2 = is1 * ( 1 + vos/vt ) ;
gm1 = ic1/vt ;
rc2 = rc1 = 1 / ( 2*M_PI*f0dB*c2 ) ;
re1 = re2 = (beta1+beta2) / (beta1+beta2+2) * (rc1 - 1/gm1) ;
c1 = c2 / 2 * tan(delta_phi) ;
/* Vcc is defined as being a positive value & Vee as a negative value */
rp = vcc-vee) * (vcc-vee) / (pd - fabs(vcc*2*ic1) - fabs(vee * iee) ) ;
ga = 1/rc1;
cmrr = pow(10.0,cmrr/20.0); /* convert from dB to decimal */
gcm = 1 / (rc1*cmrr) ;
ro1 = rout_ac ;
ro2 = rout - ro1 ;
gb = avd * rc1 / ( r2 * ro2 ) ;
ix = ( 2 * ic1) * gb * r2 - isc ;
isd1 = isd2 = ix * exp( -ro1*isc/vt ) ;
rc = vt / (100*ix) * log ( ix/isd1 ) ;
gc= 1/rc;
vc = fabs(vcc) - vout_p + vt*log(isc_p/isd3) ;
ve = fabs(vee) + vout_n + vt*log(isc_n/isd4) ;
```

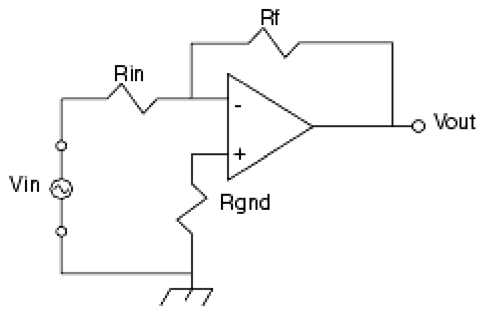
Bias Circuitry

The opamp model, fully defined and model parameters extracted, can now be used as a functional circuit block. This requires that the opamp be biased with external supply voltages and circuit configuring components. This was demonstrated previously through the use of a test circuit. The test circuit implemented forms an inverting amplifier using an input and feedback resistor with the opamp. This results in a complete functional circuit whose performance can be studied in more detail. The test circuit definition and the resulting equivalent circuit are shown in the following 2 figures.

Test Circuit Definition to Form an Inverting Amplifier

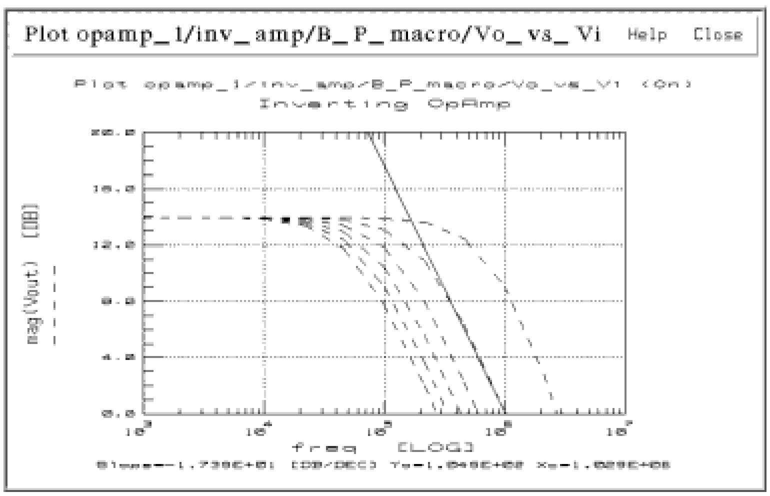
```
* Inverting Amplifier
.SUBCKT  inv_amp 1 2 3 4 6 7
X1  2 3 4 6 7  opamp_1
Rf    6  2  10K
Rin   2  1  2K
Rgnd  3  0  1.0m
.ENDS
```

Equivalent Schematic of Opamp in Inverting Amplifier



The plot in the following figure illustrates how this functional circuit can be studied. The opamp circuit has been simulated with the compensation capacitance used as one of the sweep parameters. The AC voltage gain is plotted versus frequency with steps of different values of C2. The diagonal line is the break point between process limitations and circuit limitations. To increase frequency response, the internal capacitor on the chip (nominal 30pF) would have to be reduced. To force earlier roll-off, more external capacitance can be added. This analysis indicates the range of gain-bandwidth product available for different values of capacitance. Similar analyses can be performed for any area of this circuit's operation.

AC Opamp Response vs C2 Capacitance



References

1. G.R. Boyle, B.M. Cohn, D.O.Pederson, J.E. Solomon. *Macromodeling of Integrated Circuit Operational Amplifiers,* IEEE Journal of Solid-State Circuits, Vol. SC-9, No. 6, December 1974.

Printing and Plotting

This section contains procedures for definition and creation of graphic displays and textual reports and describes printing and plotting from IC-CAP.

You can send output to printers and plotters as well as to a file in a variety of formats. When printing to a file, the format of the file is determined by the current output device and the file is saved in the current project directory.

You can connect any output device that is supported by your operating system. To connect additional printers and plotters, and select a default printer, choose the appropriate method for your platform:

- WindowsXP-Start menu > Settings > Printers
- UNIX-Choose File > Print Setup

The basic Print commands are summarized next:

- Use the Print command to print the contents of the current window
- Use the Print Setup command to establish a default printing configuration, although you can modify it at the time of printing

Creating an IC-CAP Plot

This section contains procedures for definition and creation of graphic displays and textual reports. The MOSFET model supplied with IC-CAP, *nmos2*, contains a variety of plotted data sets and is used as the example in this section.

A plot is a graphical representation of data that gives a quick view of measured and simulated data, making it easier to spot any differences between the two. Using IC-CAP's plot analysis features, you can identify a data point or slope of a curve easily.

If you need to get raw data in the form of numbers rather than a graph, you can view such data in a list form that can be sent to a printer.

The general procedure for creating a plot is:

- Open the setup and add a plot.
- Assign a data set to a graph axis.
- Specify a report type and an axis type.
- Put in a header and footer if necessary.
- Open a plot window.
- Rescale the plot if necessary.
- Analyze the plot.
- Print a hardcopy of the plot.

> **Note**
> The term data set means either Input, Output, or Transform, as used in *Creating and Running Macros* (prog).

Because the print methods vary significantly between UNIX and the PC, this section describes printing from these platforms separately. See the appropriate section for your platform:

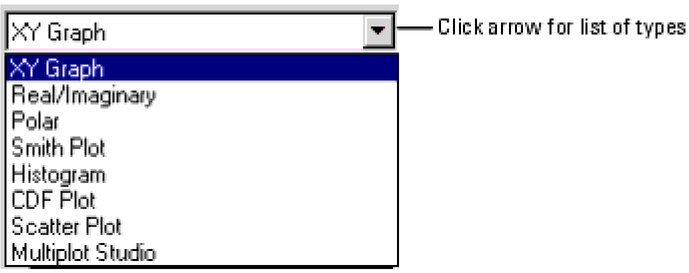
- Printing from UNIX
- Printing from the PC

Adding a New Plot

The first step in creating a plot is to add a plot.

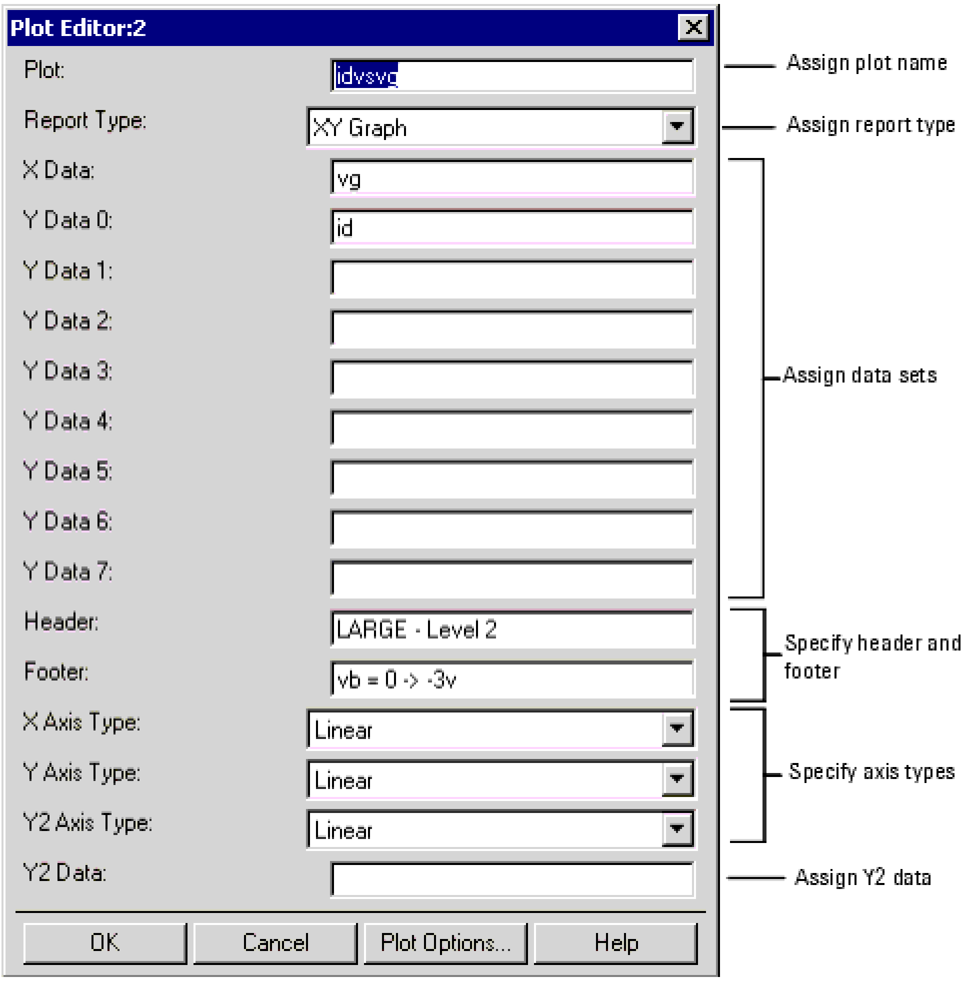
To create a new plot:

1. In the DUTs-Setups folder, select the setup.

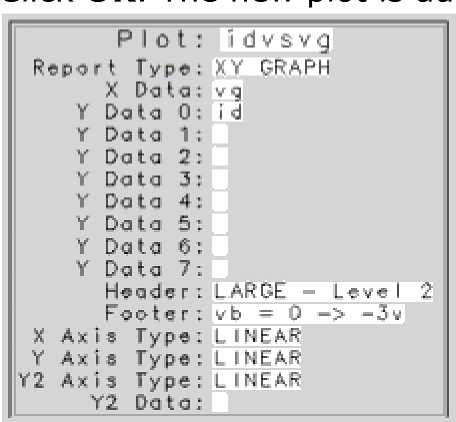

2. Then select **Plots**.
3. Click **New**.
4. In the Plot Editor, enter a name for the new plot.
5. Select one of the Report Types from the drop-down list.


6. The required fields change to reflect the selected report type. Fill in the necessary fields for the selected report type. For details, see Defining a Plot for Specific Data and Defining Axis Types and Data Sets.

> **Note**
> You can specify an alternate name for a plot legend by adding two exclamation marks (*!!*) followed by the alternate name after the real data item. For example, the following statements label the X data as *vin* and the Y data as *calc_a* :
> ```
> X Data : /a/really/really_long_path/vin !! vin
> Y Data 0 : (a + complicated)*expression !! calc_a
> ```



7. Click **Plot Options** to define plot options. See Setting Plot Options.
8. Click **OK**. The new plot is added to the Plots folder.



Defining a Plot for Specific Data

The following 8 tables list definitions for plot data types. Certain plot characteristics can be set with system variables. When a variable applies to a specific plot type, the variable is listed in the table for that type. Variables that apply to any plot are listed in Plot Characteristics Variables.

Defining a Plot for XY Data

Data Type	Definition
X Data:	The Input name
Y Data 0:	The Output name (or expression that is a function of the outputs)
Y Data 1-7:	Names of additional outputs sharing the same scale as Y0
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph
X Axis Type:	X Axis Type (Linear, Log 10, and dB) †
Y Axis Type:	Y Axis Type (Linear, Log 10, and dB) †
Y2 Axis Type:	Additional Y axis type used for showing another data set with a different magnitude or domain. For example, use the Y axis for gain and the Y2 axis for phase.
Y2 Data:	The name of a data set to display with a different scale.

† dB axis type represents 20 × log10(x), not 10 × log10(\x).

Defining a Plot for Real/Imaginary Data

Data Type	Definition
Sweep Data:	The Input data set name
RI Data 0:	The Output data set name or expression that is a function of the outputs
RI Data 1-7:	Additional data sets or expressions
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph

Defining a Polar Chart

Data Type	Definition
Sweep Data:	The Input data set name
Polar Data 0:	The Output data set name or expression that is a function of the outputs
Polar Data 1-7:	Additional data sets or expressions
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph

Defining a Smith Plot

Data Type	Definition
Sweep Data:	The Input data set name
Smith Data 0:	The Output data set name or expression that is a function of the outputs
Smith Data 1-7:	Additional data sets or expressions
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph

A histogram depicts either the frequency or the relative frequency of data distribution. Typically a histogram is used to identify both the ranges around which most of the data converges and the outliers of the data set. In a histogram, the X-axis displays the value ranges while the Y-axis plots the frequency or relative frequency for each range. A histogram is used for the visualization of data for a single variable.

Defining a Histogram

Data Type	**Definition**
Data-set:	The Input data set name
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph
Variables for setting Plot Characteristics	
HISTOGRAM_NUM_BINS	Number of bins in the histogram plot. Default is 10.
HISTOGRAM_NORMALIZATION	Can be set to either TRUE or FALSE. Setting it to TRUE will normalize the histogram. Default is FALSE. NOTE: definition of the histogram normalization: Assume Variable *X* indicated the width of the histogram bin, Variable *n* indicated the number of the histogram bins. Variable Y_i indicated the number of samples for the histogram bin with index of *i* Variable *normalize_*Y_i indicated the normalized number of samples for the histogram bin with index of *i* Then we have, $normalize\_Yi = (Yi)/\left(\sum_{i-1}^{n}(X \times Y_i)\right)$
HISTOGRAM_GAUSSIAN_FIT	Can be set to either TRUE or FALSE. Setting it to TRUE will draw the Gaussian curve fitted to the histogram. Default is TRUE.

A cumulative density plot depicts either the cumulative frequency or the cumulative relative frequency of data distribution. Typically a cumulative density plot is used to track variations between contemporaneous observations by highlighting changes in mean levels. In a cumulative density plot the X-axis displays the value ranges while the Y-axis plots the cumulative frequency or cumulative relative frequency for each range.

Defining a CDF Plot

Data Type	**Definition**
Data-set:	The name of an Input data set
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph
Variable for setting Plot Characteristics	
CDF_ERROR_FIT	Can be set to TRUE or FALSE. TRUE will draw the Gaussian curve fitted to the Cumulative plot. Default is TRUE.

A scatter plot depicts the distribution of pairs of values on a rectangular, 2-dimensional plane. A scatter plot is the most effective method of analyzing the correlation between two variables. Values for one variable are tracked on the X-axis while values for the other variable are tracked on the Y-axis.

Defining a Scatter Plot

Data Type	Definition
X Data:	The X-axis parameter name
Y Data:	The Y-axis parameter name
Header:	Annotation that appears above the graph
Footer:	Annotation that appears below the graph
X Axis Type:	X Axis Type (Linear, Log 10, and DB)
Y Axis Type:	Y Axis Type (Linear, Log 10, and DB)
Variable for setting Plot Characteristics	
SCATTER_CONTOURS	Can be set to either TRUE or FALSE. Setting it to TRUE will draw the contours. Default is TRUE.
SCATTER_NUM_SEGMENTS	Number of segments used in drawing the contours of the plot. Default is 1500. Using a larger number results in an accurate contour, but takes more time.

Defining a Multiplot Studio

Variable	Definition
# of Plots	Number of plots displayed in the Multiplot. The number can be from 1 to 100. The default is 4.
Plot 1 - 100:	Name of each plot.
# of GUIs:	Number of GUI regions. The number can be 0 to 8. The default is 0.
GUI 1 - 8:	Name of each GUI region.
GUI 1- 8 Location:	Location on the Multiplot display for each GUI region. The location can be upper left (UL), upper right (UR), upper (U), left (L), right (R), bottom left (BL), bottom right (BR), or bottom (B).
Orientation:	Orientation of the Multiplot. The orientation can be vertical (V) or horizontal (H). The default is horizontal.
Plots Per Row:	Number of plots in each row. If the number is 0, the plots will be arranged so that the number of rows and columns are equal if possible (squared layout). The default is 0.
Variable for setting Plot Characteristics	
DYNAMIC_MULTIPLOT_MODE	Sets the location where the manual scaling setting is saved for a Multiplot. If set to FALSE, the manual scaling setting for each subplot in a Multiplot is saved with the Multiplot, allowing different scaling to be saved for the same plot if opened as a single plot or on one or more Multiplot. If set to TRUE, the manual scaling setting for each subplot in a Mulitplot is saved with the subplot. In this mode, no matter where the plot is opened (as a single plot or on one or more Multiplots), it will use the same scaling information. However, if the same subplot appears on multiple Multiplots, or in multiple positions of the same Multiplot, then the settings for only the last subplot closed is saved. Default is FALSE.

Plot Characteristics Variables

Variable	Definition
ANNOTATE_AUTO	Sets a flag to enable or disable automatic annotation update upon data changes. Default is No.
ANNOTATE_CSET	Sets a *Starbase* character set to be used for annotation texts. Default value is determined by /usr/lib/starbase/defaults.
ANNOTATE_FILE	Sets a file name from which a plot reads in an annotation text. Default is no file to read.
ANNOTATE_MACRO	Sets a macro name that is executed by a Plot for generating an annotation text. Default is no macro to execute.
CHECK_PLOT_MATCH	Lets IC-CAP check if a given XY pair belongs to the same Setup. If No, potentially mismatched XY pair can be shown in a tabular format with Display Data. Default is Yes.
DASH_DOT	Sets the number of data points at which a simulated line changes from a dashed to a dotted line. Used in Plot. Default value is 32.
FIX_PLOT_SIZE	If Yes, Plot windows open using the size specified by GWINDX and GWINDY. If No, they open using the last displayed size. Default is No.
GWINDX	Sets the initial Plot window horizontal size in 1/100mm †. Used in Plot. Default value is 12500.
GWINDY	Sets the initial Plot window vertical size in 1/100mm †. Used in Plot. Default value is 9000.
IGNORE_PLOT_LOC	If Yes, Plot windows open using the X windows system configuration. If No, they open using the last displayed location. Default is No.
MINLOG	Can be set to a real value. Defines the value to be used in a LOG plot, if data point value is zero or negative. Default is 10e-18.
OFFSCREEN_PLOT_LINE_ WIDTH	Sets the line thickness used when drawing a plot to an HPGL file or HPGL printer. OFFSCREEN_PLOT_TRACE_LINE_WIDTH will override this value for the traces on a plot. Default is 0.
OFFSCREEN_PLOT_TRACE_LINE_WIDTH	Sets the line thickness used when drawing the traces of a plot to an HPGL file or HPGL printer. Default is 0.
PLOT_LINE_WIDTH	Sets the line thickness used when drawing a plot. PLOT_TRACE_LINE_WIDTH will override this value for the traces on a plot. Default is 1.
PLOT_TRACE_LINE	Sets whether trace line is drawn or not. If defined as Yes, the trace lines are drawn. If defined as No, the trace lines will not be drawn, and instead markers will be drawn. Default is Yes.
PLOT_TRACE_LINE_WIDTH	Sets the line thickness used when drawing the traces of a plot. PLOT_TRACE_LINE_WIDTH will override this value for the traces on a plot. Default is 1.
RETAIN_PLOT	When Yes and Auto Scale is off, plot is not erased when updated to allow overlay of curves if the X server has *backing store* capability. Default is No.
RI_GRAPH_SYMMETRY	When defined as Yes, the plot title is displayed. If defined as No, the plot title is not displayed. Default is Yes.
SHOW_GRID	When No, plot eliminates XY grids and leaves tics. Default is Yes.
USE_PLOT_LOOKUP	Lets IC-CAP perform auto-lookup of X data from each Y data. Another way to disable auto-lookup is to use an arbitrary expression for an X data. Default is Yes.

† On-screen size differs depending on screen dpi calculated with OS-reported screen width/height.

Editing a Plot

To edit an existing plot definition:

1. Open the Plot Editor:
    - From the plot window, click **Options** > **Edit Definition** or right click on the plot then choose **Edit Definition**.
    - From the DUTs-Setups folder, select the setup. Then select **Plots**. Now double-click the plot table you want to edit or select the plot table and click **Edit**. Alternatively, you can edit directly in the plot table. See Table and Text Editors.
2. Change the fields as needed and click **OK**. The plot definition is updated.

Notes:

- If the plot window is open when you click OK, the plot window closes then immediately reopens with the new definition. The exception is if you were editing the definition of a sub-plot in a Multiplot window. In that case, the Multiplot window refreshes with the updated definition. If the selected plot was also displayed in a single plot, the single plot closes and does not reopen.
- In a Multiplot window, if you right click on a plot to open the Plot Editor and change the plot's name, when you click OK the plot disappears from the Multiplot window.

Tips:

- Use Display Plot to view the plot window for an individual, selected plot.
- Use Display All to view plot windows for all defined plots.
- Use Close All to close all open plot windows.
- For all plots except Multiplot, use View to see data in a tabular format.

Defining Axis Types and Data Sets

In the example definition, both X axis and Y axis are defined as LINEAR. You can change this type to either LOG10 or DB. Although the example does not need to have a log Y axis, try changing it:

1. Select the example plot, **idvsvg**. Click **Edit**.
2. In the Y Axis Type field, toggle to **LOG10** and click **OK**.
3. Click **Display Plot** to view the graph. The graph *idvsvg* is updated with a logarithmic Y axis.

> **Note**
> The XY GRAPH is not a convenient means for viewing complex numbers. If you want to make a graph with complex numbers, use either the RI GRAPH or the POLAR GRAPH. These graphs take one sweep data set, such as frequency, and generate a Real/Imaginary graph or a Polar graph.

Y2 Data

Another Y axis, called Y2 data, is available for XY GRAPH. The Y2 data axis is useful for showing another data set with a different magnitude or domain. For example, in a gain and phase versus frequency graph you can use the Y axis for gain and the Y2 axis for phase. The Y2 axis always requires one or more Y data sets because a slope of a Diag Line is derived from the left axis values.

The program assumes that Y2 data shares the same setup with the X data, so the program never looks up the Y2 data and always draws against the X data in the plot definition.

Expression

An expression can be entered into the data set fields described above, just like any other fields where an expression is allowed. For example, you can enter a natural log of *id* as *log(id)* in the Y Data 0 field. The calculated data set belongs to the setup where this plot definition exists.

> **Note**
> A scalar value is expanded to an array of the constant value so that it shows up as a straight line on a graph. For more information on functions, refer to *Using Transforms and Functions* (prog).

Multiple X Data

IC-CAP allows plots to have multiple XY pairs in a single graph by looking up the same X data name in another setup. This is useful when comparing multiple XY pairs with different X ranges.

Auto Lookup

Each Y data set is examined for its corresponding X data set. This feature, called Auto Lookup, takes the X data name from the plot definition and searches this name in the setup that has this particular Y data in question. If there is no such X data in that setup, then the X data in this plot definition is used.

For example, the next definition draws two different sets of *id vs vg* curves based on large and short setups so that one set represents id vs vg of nmos2/large/idvg setup while the other set shows that of nmos2/short/idvg. The Y data names do not have to be the same. This feature supports all the report types.

Id vs Vg #1

Report Type	**XY GRAPH}**
X Data	/nmos2/large/idvg/vg
Y Data 0	/nmos2/large/idvg/id
Y Data 1	/nmos2/short/idvg/id

If this plot exists in the nmos2/large/idvg setup, then the next definition is good enough to show the same curves. In both cases, the number of data points and points per curve may vary among Y data sets because each Y data is shown against its corresponding X data from their home setup.

Id vs Vg #2

Report Type	**XY GRAPH**
X Data	vg
Y Data 0	id
Y Data 1	nmos2/short/idvg/id

The tabular report drops each Y data set that does not share the X data of the plot definition, because representing multiple, possibly independent X and Y data pairs in a simple 2-dimensional table is difficult.

For example, the plot definition in the second table shows the *vg* and *id* of nmos2/large/idvg setup only. When you must show all data sets that do not share the same X data, set the system variable CHECK_PLOT_MATCH to *No.*

Calculated Data

If an expression is entered into a plot definition, then the calculated result belongs to the setup where the plot definition exists. Therefore, when using a transform, you must calculate data within its home setup.

For example, the next definition based on the previous example gives the wrong curves for Y Data 1 because the *id* in the short setup is drawn against *vg* in the large setup. If the data size does not match between the X and Y data sets, the program issues an error message.

Calculated Id

Report Type	**XY GRAPH**
X Data	vg
Y Data 0	id
Y Data 1	nmos2/short/idvg/id*2

Disabling Lookup

You can disable the Auto Lookup so that all the Y data are drawn against the same X data even if these Y data sets do not belong to the same setup with the X data. To turn off the Auto Lookup set the system variable USE_PLOT_LOOKUP to *No*.

Displaying Raw Data

You can view the raw data for an input, output, transform, or plot (except Multiplot) by selecting it and clicking View (in their respective folders). The data format displays the data point number, data index, and the real and imaginary part of the data.

- Point shows the number of sweep data point starting from 0 (zero).
- Index represents a pair of subscripts for a matrix data, such as S-parameters. These are in row, column format.

The actual data points for an input, output, or transform are shown in real and imaginary format represented in the header by *R* and *I*.

The data format for plots follows the report type of the plot itself. When the report type is *XY GRAPH* only the real part of the data is displayed.

- Pnt/cv shows two digits, as discussed in Marking a Point.
- M indicates measured data and S indicates simulated data.
- C denotes common data that is shared between measurements and simulations (for example, an Input).

The tabular data of a plot shows the Y data that belongs to the setup of this plot. Foreign Y data is not displayed because of the different X data. See Y2 Data.

Displaying Plots

You can open a plot window to view your measured and simulated data (see the following figures). The currently-defined graphs are listed in the Plots folder in each setup. You can open one or more plot windows at a time and each display appears in a separate window. For all plots except Multiplot, you can view the same data at the same time on a graph and in a tabular format.

Measured data is displayed as a solid line; simulated data is displayed as small squares by default. Plots are automatically updated each time a measurement or simulation is performed. After an extraction and subsequent simulation, you can view the plots for agreement between measured and simulated data.

- To view the plot window:
  - For an individual, selected plot, click **Display Plot** 

  - For all plots defined in a setup, click **Display All** 

- To see data in a tabular format for all plots except Multiplot, click **View**. 

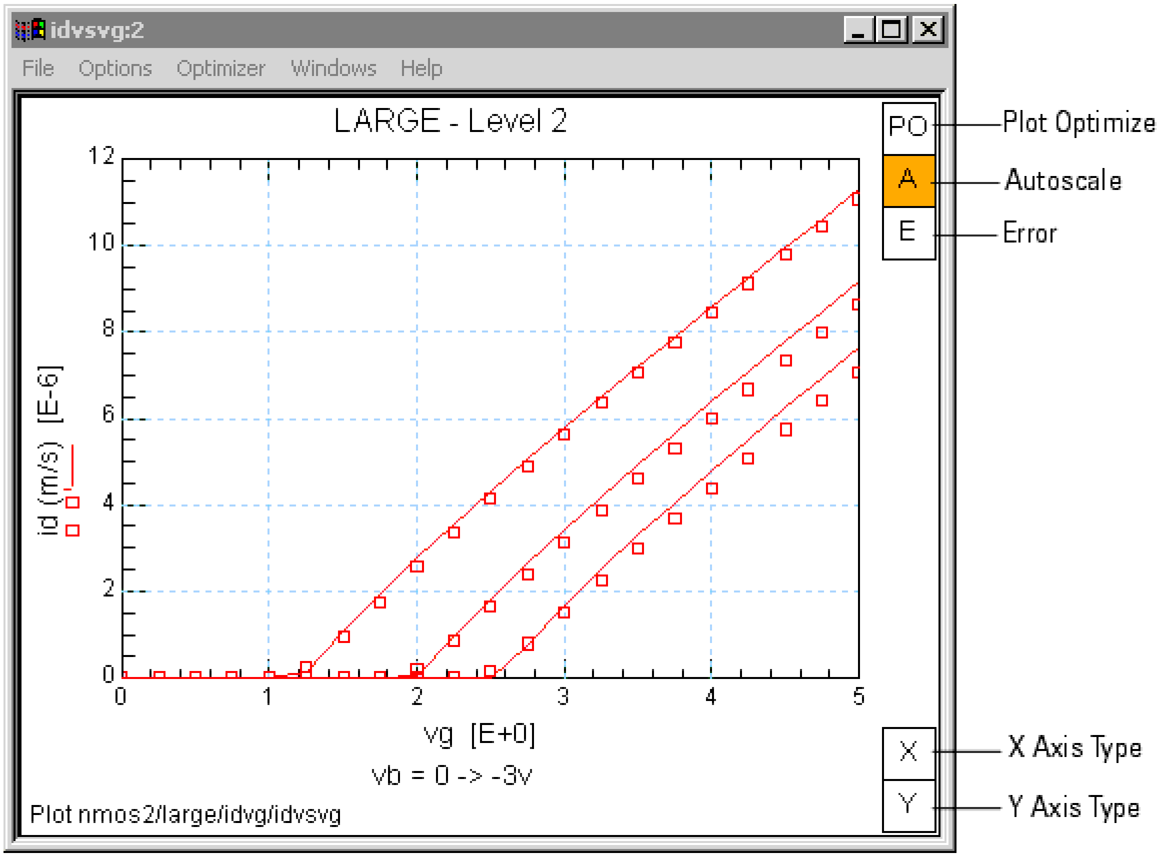
- To close all open plot windows, click **Close All**. 

> **Note**
> Alternatively, you can display a plot by selecting a plot definition and clicking the right mouse button. Then choose an option from the menu.
> If you want to display or close plots while a different folder is open, use the tool bar buttons at the top of the Model window: Display Plots (Setup), Close Plots (Setup), Display Plots (DUT), Close Plots (DUT).

Area Tools

To display Area Tools, choose *Options* > *Area Tools* or right click then choose *Graphic* > *Area Tools*. The following figure shows a Plot window with Area Tools displayed.

Example Plot Window



The *PO* (Plot Optimizer) area tool performs the same function as the *Optimizer > Enable/Disable Plot* menu pick. When enabled, the PO area tool is blue and the plot has a blue border. For additional information about the Plot Optimizer, see *Using the Plot Optimizer* (sim).

The *A* (Autoscale) area tool performs the same function as the *Options > Autoscale* menu pick. See Scaling.

The *E* (Error) area tool performs the same function as the *Options > Error* menu pick. See Defining and Displaying Errors.

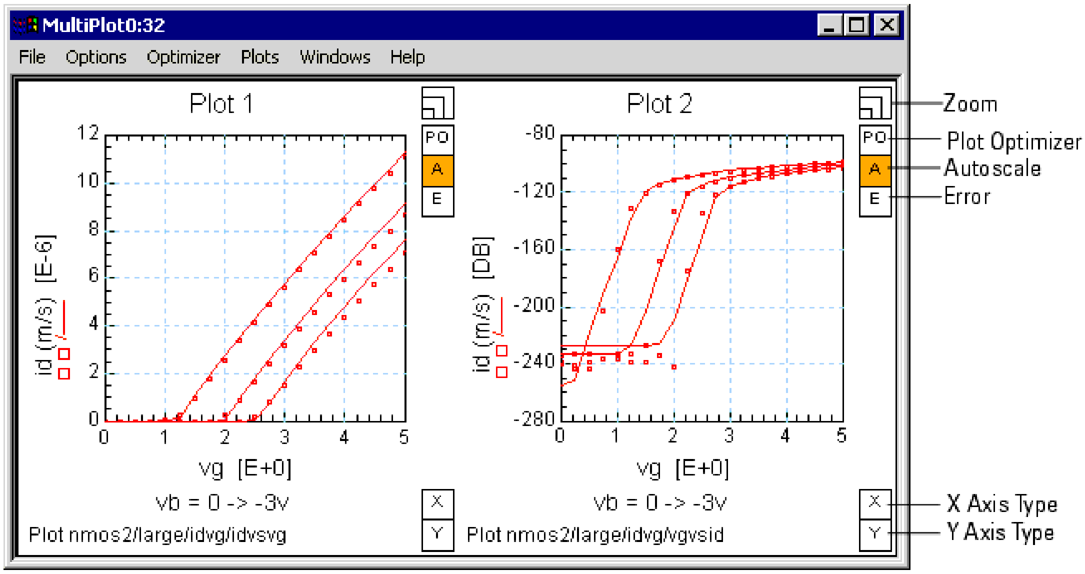
The *X* (X Axis Type) area tool enables you to toggle through the available X axis types from the plot window. To view the current selection, position and hold the cursor over the area.

The *Y* (Y Axis Type) area tool enables you to toggle through the available Y axis types from the plot window. To view the current selection, position and hold the cursor over the area.

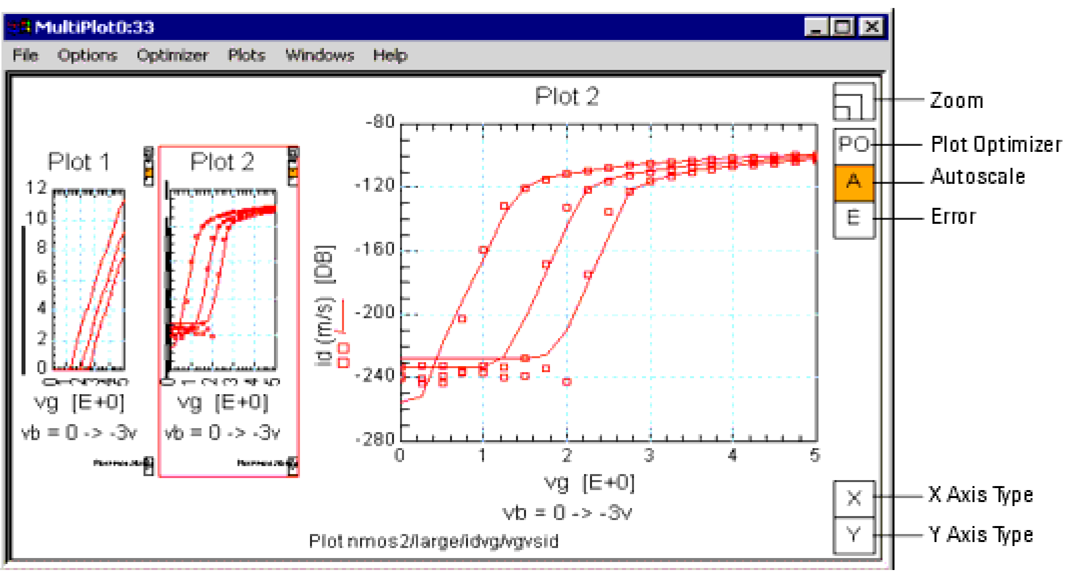
The *Y2* (Y2 Axis Type) area tool enables you to toggle through the available Y axis types from the plot window. This area tool is only shown when a Y2 trace is present.

Multiplot windows also have a Zoom area tool. The Zoom area tool enlarges the selected plot. A red border indicated which plot is selected. When Zoom is enabled, you can change which plot is enlarged simply by clicking on a different plot. In addition, the Plots menu enables you to zoom in or display a full page plot. Example Zoomed Multiplot Window shows a 2 plot Multiplot window with Zoom enabled.

Example Multiplot Window



Example Zoomed Multiplot Window



Scaling

Autoscale

Autoscale can be toggled on and off by either:

- Choosing *Options* > *Autoscale*
- Right clicking on the plot then choosing *Scaling* > *Autoscale*
- Clicking the plot's *A* area tool.

If a check mark is before the *Autoscale* menu item or the plot's *A* area tool is orange, Autoscale is on. When Autoscale is on, the axes will autoscale so all data is visible. When Autoscale is off, the state of this menu item is saved to *.mdl* files. For example, if you save a plot (or a model containing the plot) with Autoscale turned off, the plot retains the setting, and opens in the Autoscale OFF state the next time the plot is opened.

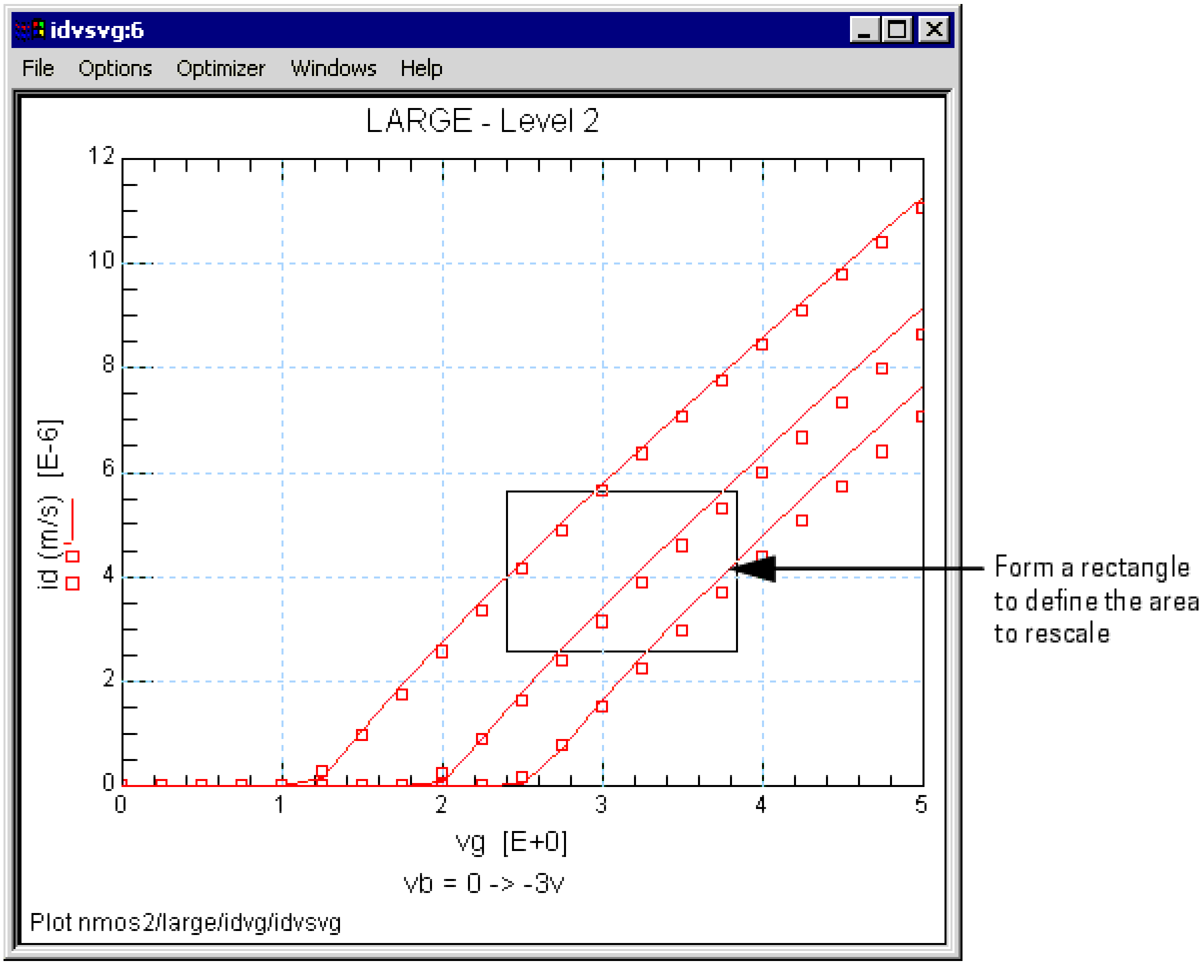
Axes scaling is controlled by settings you choose in the Manual Rescale dialog box (Options > Manual Rescale). If you have a new plot, or a plot that has never had the scaling modified, turning Autoscale off locks the current settings in place. Any future toggling of Autoscale ON/OFF will switch between full scale, and this remembered setting. The remembered setting can be further modified by using the Options > Manual Rescale or Options > Set Scale menu choices.

Rescale

You can zoom into a selected area of a plot by drawing a box around a portion of the plot, then either:

- *Choose Options > Rescale*
- Right click on the plot and choose *Scaling > Rescale*
- Click the *A* area tool
- In a Multiplot window, select a plot then choose *Plots > Selected Plot Menu > Scaling > Rescale*

Also, you can pan across the plot by selecting a single point on the plot then choose *Rescale*. The plot moves so the selected point is at the display's center. When Autoscale is on, the rescaling is temporary. The next replot restores the plot to full Autoscale.

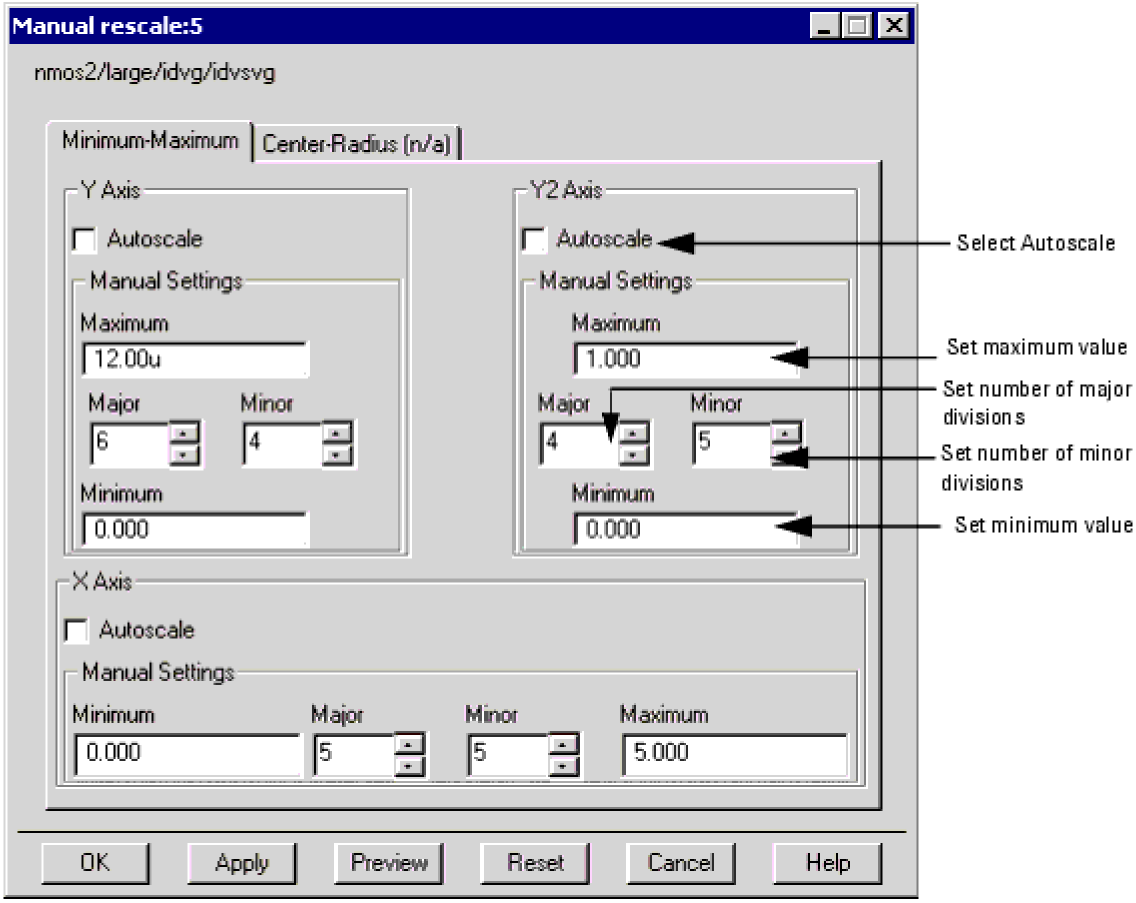


Set Scale

The *Options > Set Scale* menu item is a shortcut to establish Manual Rescale settings. You can also access this menu item by right clicking on the plot then choosing *Scale > Set scale* or from a Multiplot window by first selecting a plot then choosing *Plots > Selected Plot Menu > Scaling > Set scale*. Whether you are in autoscale mode or had performed a Rescale to zoom in on a region of the plot, choosing Options > Set Scale sets the Manual Rescale dialog box to the plot's current scaling values (e.g., X min, X max, Y min, Y max, etc.).

Manual Rescale

The *Manual rescale* dialog box enables you to fully describe all three axes of XY plots in terms of minimum value, maximum value, number of major divisions, and number of minor divisions. You can access this dialog box by choosing *Options > Manual rescale* or right clicking on the plot then choosing *Scaling > Manual rescale.* In a Multiplot window, you can access this dialog by selecting a plot then choosing *Plots > Selected Plot Menu > Scaling > Rescale*.



In addition

- You may leave any of the three axes autoscaled if you like.
- You may scale the real and imaginary axes of RI plots as well. RI, Smith, and Polar plots can be centered around a specified point with a specified radius displayed.
- You may specify the number of major and minor divisions for RI and Polar plots, but not for Smith plots.

Histogram and cumulative density plots permit scaling of the X axis and scatter plots permit scaling of the X and Y axes. Some requests may be denied due to algorithmic constraints, and in this case, the closest match is made updating the dialog with the actual values used. For example, LOG scaled axes are forced to the next decade, and the number of major and minor divisions are ignored. The number of major divisions in polar plots must be even.

Manual Rescale enables you to set the minimum to a specific value, such as .123, without rounding the value up or down. While this gives you much greater control, it can cause some problems with numbers overwriting each other, usually on the X axis. However, LOG scaled axes will still round up or down to the nearest decade.

Any settings established using *Manual Rescale*, the associated *iccap_func* command, or Options > Set Scale are saved with the plot in the .mdl file. To establish settings, choose OK or Apply. (OK also dismisses the dialog box.) To see what a plot looks like before establishing the current settings, choose Preview. To restore settings to the previously established ones, choose Reset or Cancel. To dismiss the dialog box, choose Cancel.

The settings in this dialog box update dynamically whenever a plot's scale changes. Scales change dynamically for various reasons, including when a simulation changes an autoscaled limit, or the Options > Rescale menu item is chosen. Updated settings do not change your established settings until you choose OK, Apply, or Options > Set Scale. Dynamic updating is convenient when you want to establish new settings.

Manual Rescale is fully controllable through PEL. To rescale a plot, use the iccap_func command for Plot:

```
iccap_func("Plot","<keyword>")
```

The allowed keywords include:

- *Scale Plot*
- *Scale RI Plot*
- *Scale Plot Preview*
- *Scale RI Plot Preview*

> **Note**
> The keywords Scale RI Plot Preview and Scale Plot Preview function in the same way as Scale RI Plot and Scale Plot, but the scaling will be lost on the next Replot command.

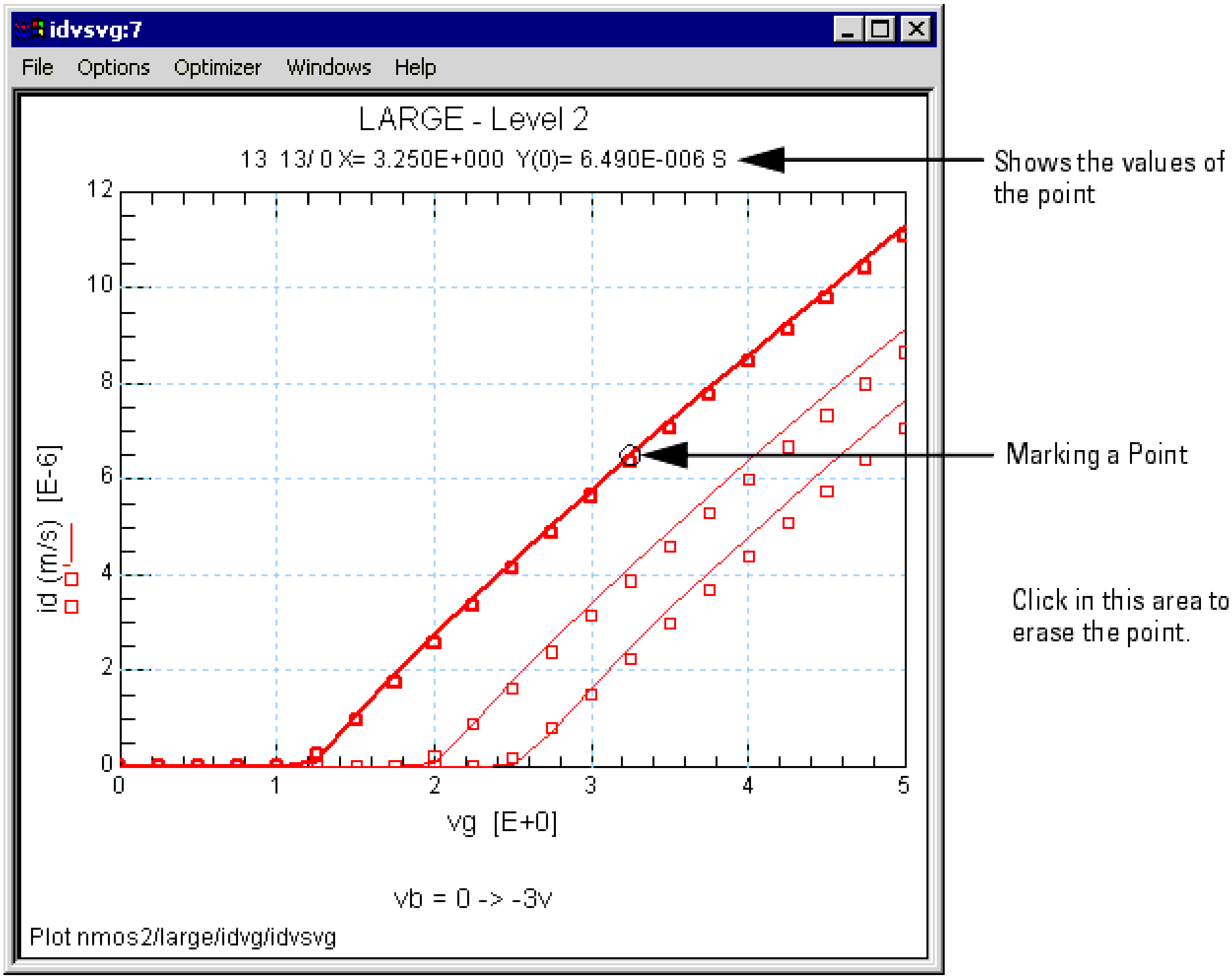
When the *iccap_func* command is used as shown above, prompts appear for the operator to enter required values for the parameters that describe the plot's scaling values. For additional information, see *Scale Plot/Scale Plot Preview* (prog) or *Scale RI Plot/Scale RI Plot Preview* (prog).

Marking a Point

You can identify the nearest data point on the graph and show its X and Y values and data point number by marking a point.

To mark a data point:

Click the Left mouse button on the graph.



To erase the circle that marks the point:

> Click the Left mouse button in the graph window, but on the outside of the graph itself. This erases the circle that marks the point, as well as its data values.

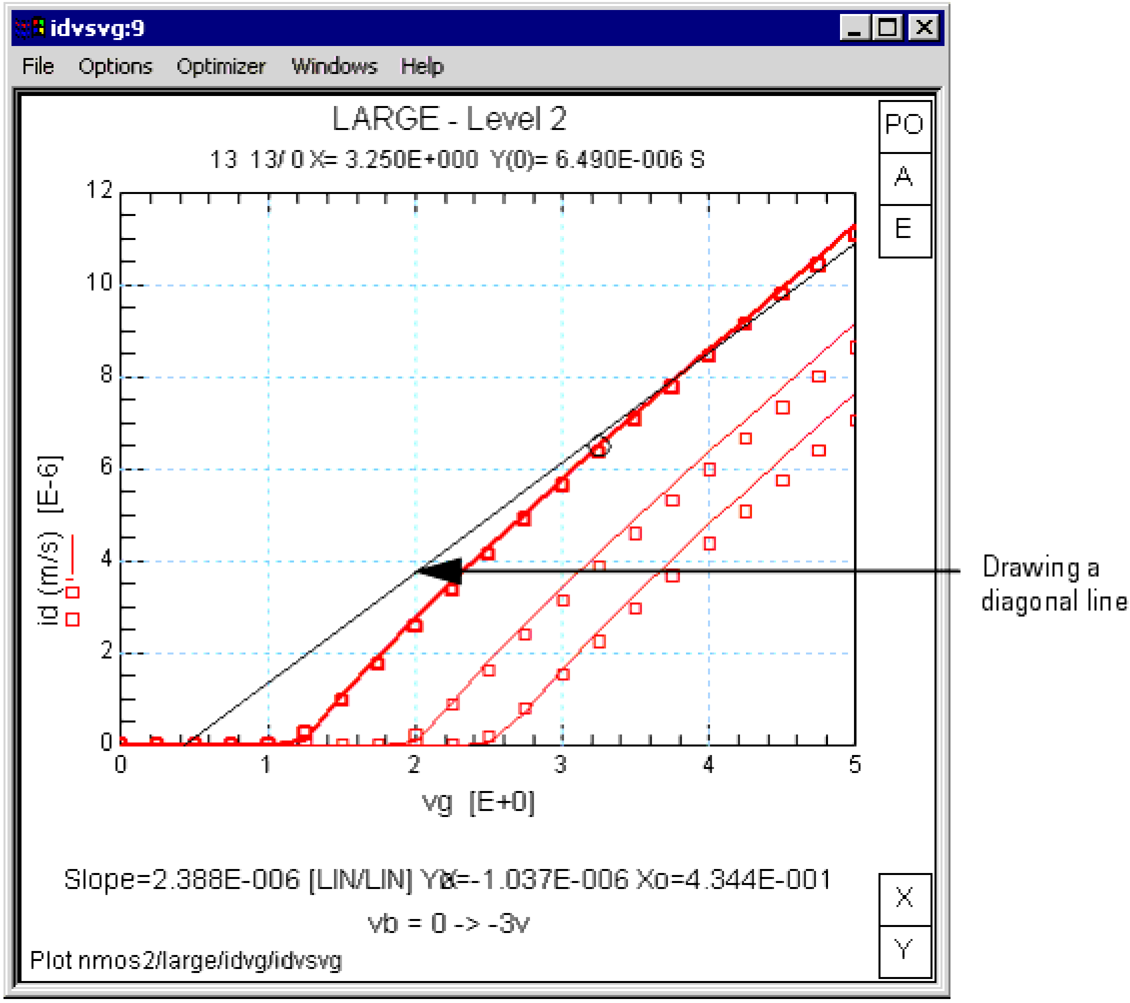
The data point number has two digits separated by a slash. The number on the left of the slash represents the primary sweep point number (sweep order 1), and the number on the right shows the secondary sweep point number (sweep order 2). A data point on Y2 axis can not be identified.

Drawing a Diagonal Line

You can draw a diagonal line connecting two clicked points and its slope, with both X and Y axis intercepts.

To draw a diagonal line:

1. Click and drag the Left mouse button from one point on a curve to a different point forming a rectangle.
2. Choose **Options > Draw Diag Line** (or from a Multiplot window, choose **Plots > Selected Plot Menu > Graphic > Draw Diag Line**).



Note
To erase the diagonal line, select **Options > Draw Diag Line**, without going through the mouse clicks forming the rectangle.

Setting Variables

When a rescale rectangle is shown on a plot, its X and Y values can be copied to system variables. Four variables (X_LOW, X_HIGH, Y_LOW, and Y_HIGH) are reserved for this purpose. You can use this feature when setting X and Y limits for optimization by specifying these variables in the Optimizer Options Table.

1. To perform *Set Variables* define these system variables at any level. Only the variables of interest must be defined.
2. With the left mouse, click and drag on a plot to form a rectangle.
3. Choose **Options > Copy to Variables** (or press C on the keyboard). From a Multiplot window, choose **Plots > Selected Plot Menu > Graphic > Copy to Variable.**

   Note
   When a Multiplot includes plots from different setups, choosing *Copy to Variable* updates the variables in the setup where the single plot was initially defined and does *NOT* update the variables in the setup for the Multiplot.

> **Note**
> The data values are taken directly from the plot. Therefore, you may need to transform those values to another form when they are used in the Optimizer Options Table. For example, if the optimization target is log(ic) and the plot shows ic versus ve with LOG10 Y axis type, the Y Bound in Optimization should be log(Y_LOW) and log(Y_HIGH).

Defining and Displaying Errors

You can display a plot's relative or absolute error based on either the entire plot or a selected region. The Error menu is only active when the Y2 trace is not occupied by another trace. In addition, only one trace of type Both can be present and if 2 traces are present, one must be of type Measured and the other of type Simulated or calculated. To view the Error menu, choose *Options > Error* or right click on the plot then choose *Error*. The Error menu contains the following choices:

- *Show Relative Error* toggles the MAX and RMS relative errors in the footer area on or off.
- *Show Absolute Error* toggles the MAX and RMS absolute errors in the footer area on or off.
- *Select Whole Plot* uses all points in the measured/simulated datasets to calculate the error.
- *Select Error Region* uses only the points within a defined region to calculate the error. To define a region, click and drag on the plot to form a box. When *Select Error Region* is selected, a green box that delimits the error calculation replaces the white box.

  > **Note**
  > A simulated/calculated data trace usually varies during tuning/simulation. Therefore once a region is selected, only a fixed number of measured points inside the green box are used to calculate the error terms.

  If *Area Tools* are enabled, you can click on the plot's *E* area tool to toggle between displaying relative error, absolute error, or not displaying any error. The E area tool is green if error calculation is enabled. To enable Area Tools, choose *Options > Area Tools*.

Setting Plot Options

Using the Plot Options dialog box, you can define trace options, plot options, text annotation, and for Multiplots, specify a PEL callback to run whenever the selected plot changes.

The Plot Options dialog box can be opened from the:

- Main window, Tools menu
- Model window, Tools menu
- Plot Editor dialog box
- Plot window, Options menu or right click menu (except from Scatter, Histogram, or CDF plots that were opened by the Statistic Package)

If the Plot Options dialog box is opened from the Plot Editor dialog box or from a plot window, a *Preview* button is displayed between the *OK* and *Cancel* buttons. To apply the current settings to the plot without closing the Plot Options dialog box, choose the *Preview* button.

To apply the current settings and close the dialog box, choose the *OK* button.

To discard all changes and close the dialog, choose the *Cancel* button.

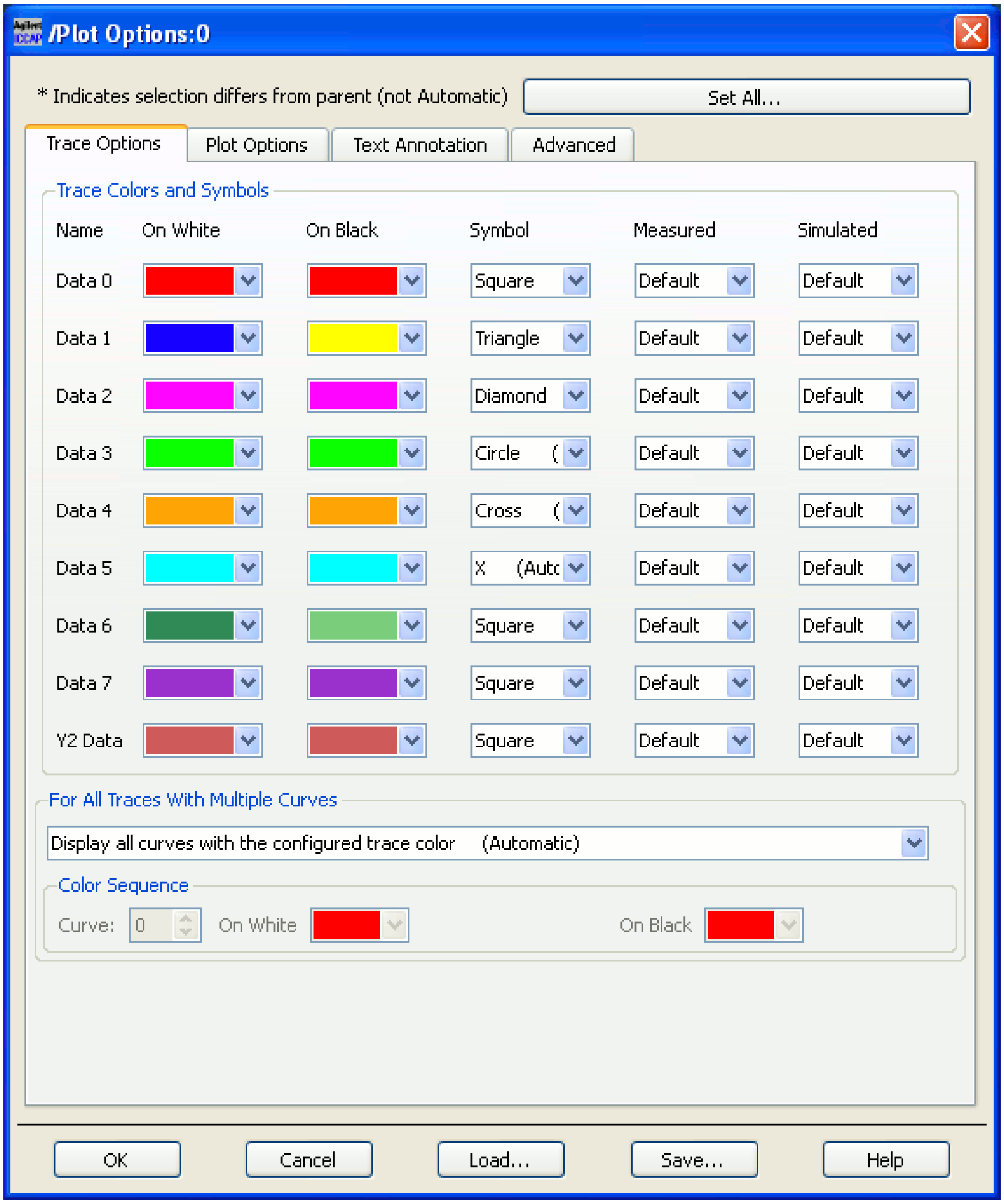
To save the current settings to a file for later use, choose the *Save* button. The *Save Plot Options* dialog box appears. Enter a file name with a *.pot* extension and choose a file location.

To load previously saved plot option settings, choose the *Load* button. The *Open Plot Options File* dialog box appears. Locate the saved file then choose Open.

Trace Options

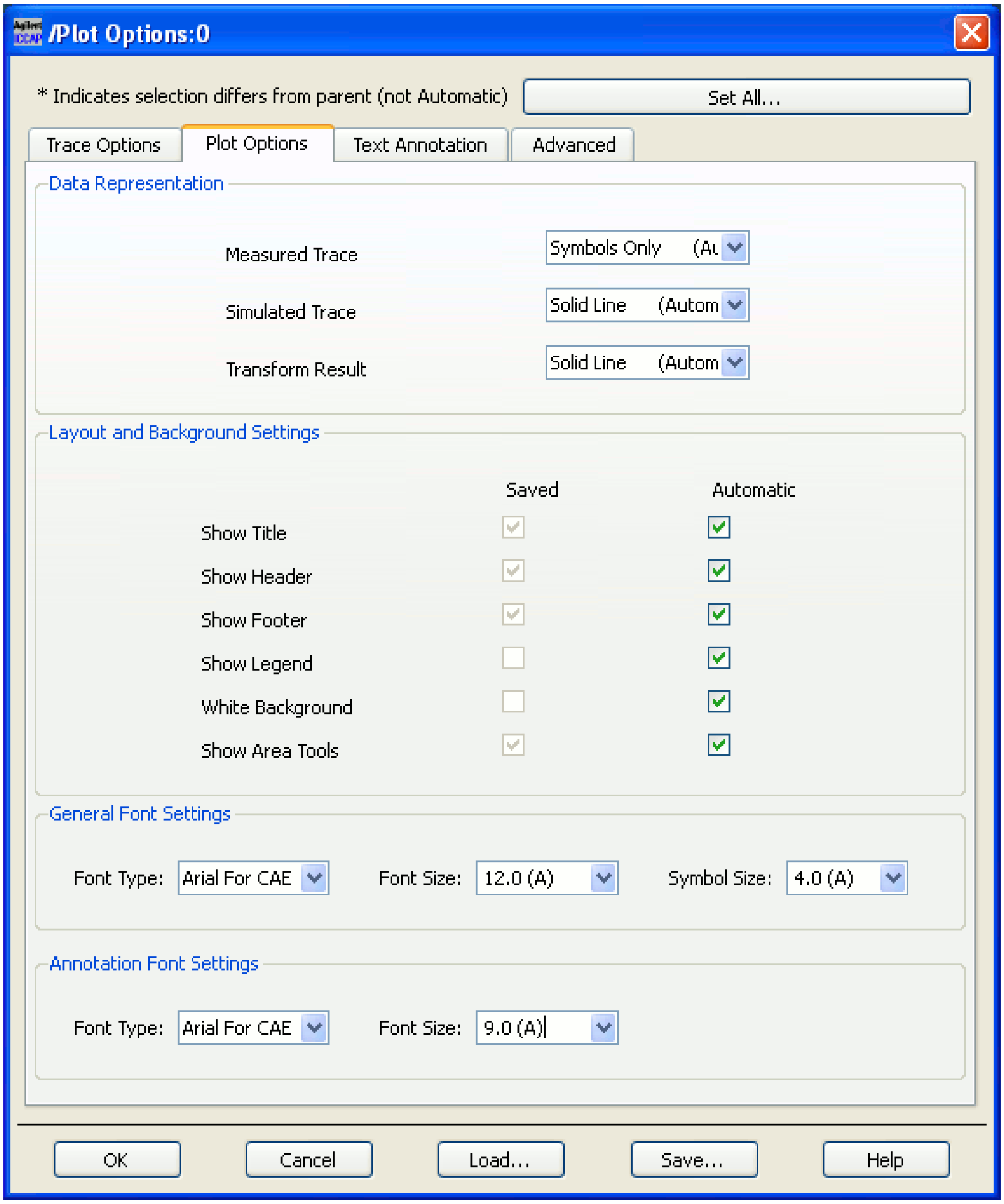
The *Trace Options* tab enables you to define trace colors and symbol shapes for all plots except Scatter, Histogram, and CDF plots. If you select Automatic, the default IC-CAP settings is used. If you select a specific color, an asterisk is display next to the color to indicate that it is not the default color.

You can also define the line types for Measured and Simulated Data. Default is a special selection, it will use the settings on plot options tab.
Take Measured Data for an example, if you choose Default, it will use Measured Trace setting in plot options tab as current selection for that Y data only, if you choose something other than Default, that Y data will use that setting for measured data.



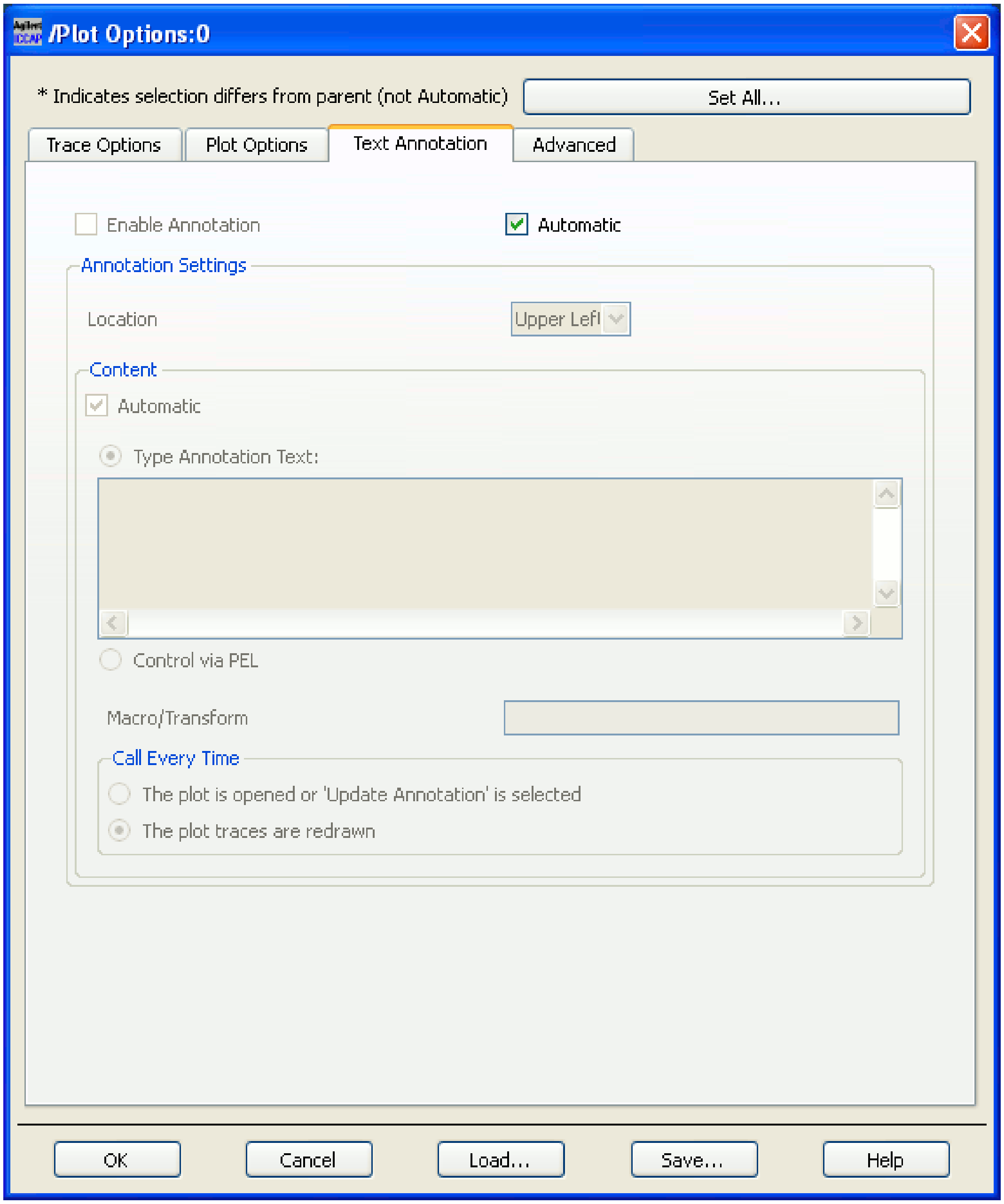
Plot Options

The *Plot Options* tab enables you to define Data Representation, Layout and Background, and Text Font. However, for Scatter, Histogram, and CDF plots, the Data Representation section is not available.



Text Annotation

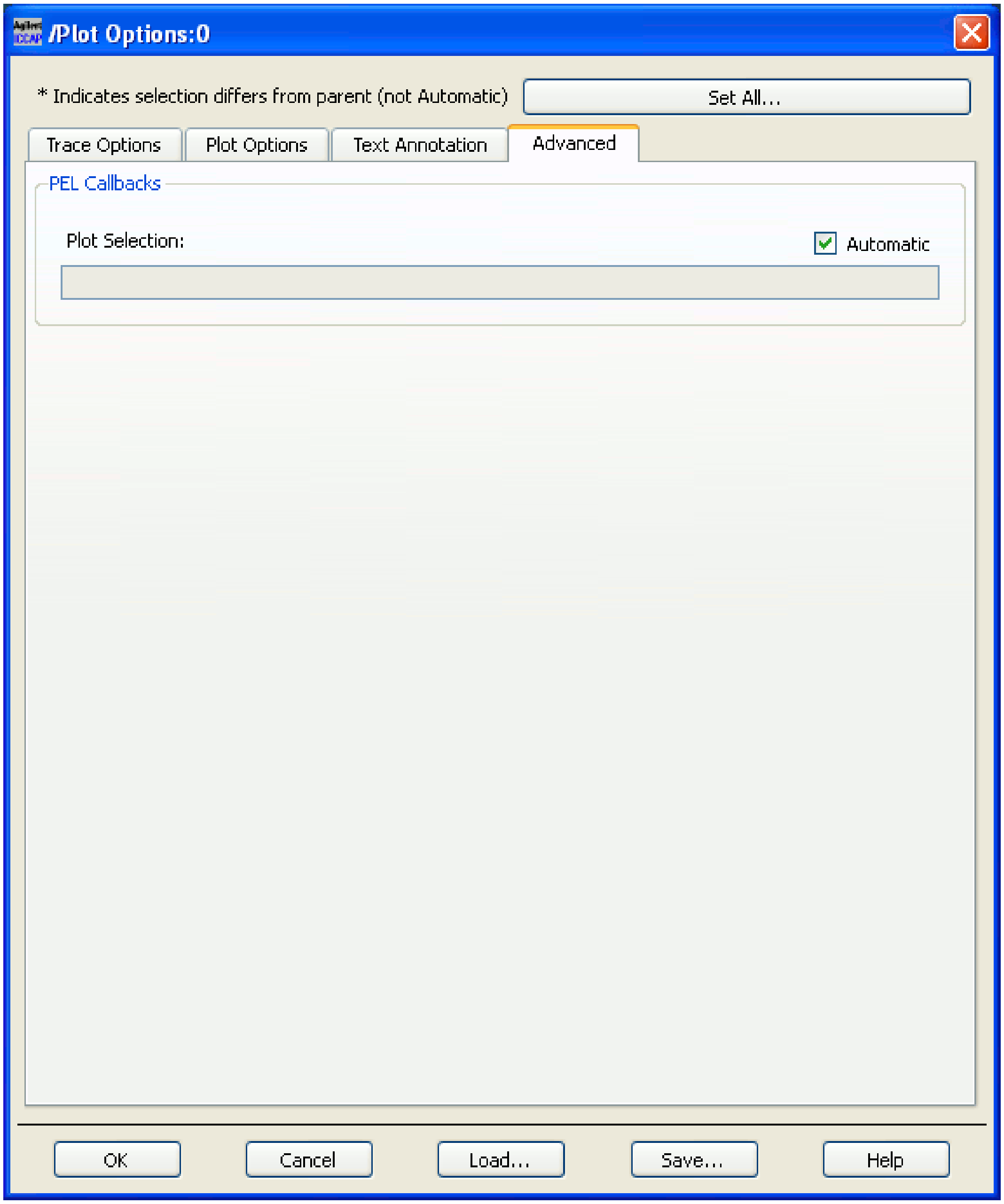
The *Text Annotation* tab enables you to add annotation text to document information such as date, lot number, simulation parameters, and so on. You can either directly enter text annotation or you can specify a PEL macro (see Annotating a Plot Using a PEL Macro).



Advanced

The *Advanced* tab enables you to specify a PEL callback (macro or transform) to run whenever the selected plot in a Multiplot changes. Note that reselecting the same plot will not invoke the callback.

To specify a PEL callback, unselect *Automatic* then type in the callback path and name. Using a relative path, the callback is found relative to the Multiplot. Therefore, the path for macros is ../../MacroName and for transforms is ./TransformName. This is true regardless of where you open Plot Options. For example, if you open Plot Options from the Model window, the relative path for a macro is still ../../MacroName.



The callback passes 3 arguments to the PEL function that are received by calling linput 3 times at the beginning of the callback. The first linput receives the name of the Multiplot where the selection is occurring. The second linput receives the new slot number selected (according to the entries on the plot definition, i.e., Plot 0 will be slot 0, Plot 2 will be slot 2, and so on). The third linput receives the old, no longer selected, plot number. A slot number of -2 indicates that no plots were selected.

Example Plot Selection Callback

```
linput "plot?",plot
linput "slot?", slot
linput "oldSlot?", oldSlot
print "newSlot=";slot;" oldSlot=";oldSlot;" Plot: ";plot
```

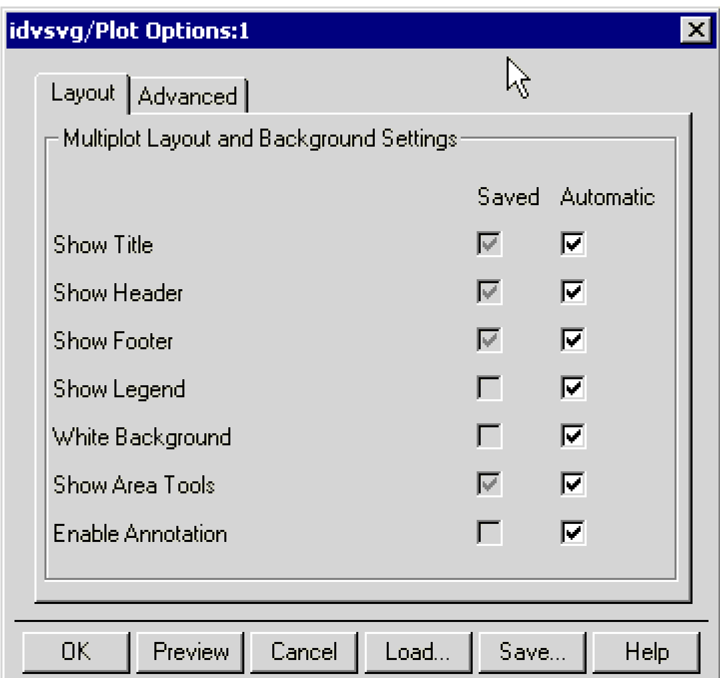
Restrictions

- This callback can only identify when the selection changes. Currently no mechanism exists that can detect when a plot is clicked on *again,* reselecting the same plot.

- Calling *iccap_func()* with "Select Plot" is not allowed during the selection callback and doing so will have no effect. Selection can occur just prior to displaying a right click menu or at the start of a click and drag. Changing the selection at these moments would be problematic, and allowing selection changes for some selections but not others can be equally problematic, so it is not allowed.
- Deleting the invoking Multiplot or the plot that is being selected is not allowed during the selection callback.
- You can modify Multiplot entries during a callback, but any call to *Replot* will do nothing during the callback. This is because changing a graph between selection and display of the right mouse menu can lead to problems.
- In general you should not change the Multiplot, its selection, or any of the child plots during a selection callback. The purpose of the callback is to allow an embedded GUI to change states in reaction to the selected plot.

Multiplot Layout

The *Layout* tab enables you to define the layout and background settings for a Mulitplot.



Setting Other Plot Characteristics

You can enable or disable several other plot characteristics that are available on the Plot window's *Options > Session Settings* menu. Although IC-CAP only retains these settings during the current session, you can save the current setting to Plot Options, or you can reset your settings back to the saved Plot Options.

- *Area Tools* turns the graph's area tools on or off.
- *Legend* turns the graph's legend on or off.
- *Text Annotation* turns the graph's text annotation on or off.
- *Title* turns the graph's title on or off.

- *Header* turns the graph's header on or off.
- *Footer* turns the graph's footer on or off.
- *Exchange Black-White* reverses the black and white settings for the graph's grid, text, and background.
- *Color* turns color on or off for the traces and markers. When *Color* is off, the traces and markers are the same color as the graph's grid and text.
- *Reset to Saved Options* resets the current session settings back to the saved Plot Options. This menu pick is not available from Scatter, Histogram, or CDF plots that were opened by the Statistic Package.
- *Save Current Settings* saves the current session's settings as the saved Plot Options. This menu pick is not available from Scatter, Histogram, or CDF plots that were opened by the Statistic Package. However, for these plots you can open the Plot Options dialog box from the IC-CAP/Main window to save the settings.

Saving a Plot Image

You can save an image of a plot with its current characteristics and size.

To save a plot image:

1. Select **File > Save Image**.
2. In the Plot Image File Name dialog box, set the location where you want to save the file.
3. Enter the file name in the form of *filename.ext* where *ext* is gif, jpg or png.

   > **Note**
   > You can save plots in various formats in Plot windows, by using the File > Save Image menu item. This feature uses ImageMagick's "Convert" program. See Help > About IC-CAP for more information. Agilent Technologies officially supports the .GIF, .JPG and .PNG formats, though .EPS, .PS, .HTML, .TIF, and other formats are available.

4. Select **OK** or **Save**. There may be some delay while IC-CAP processes the image.

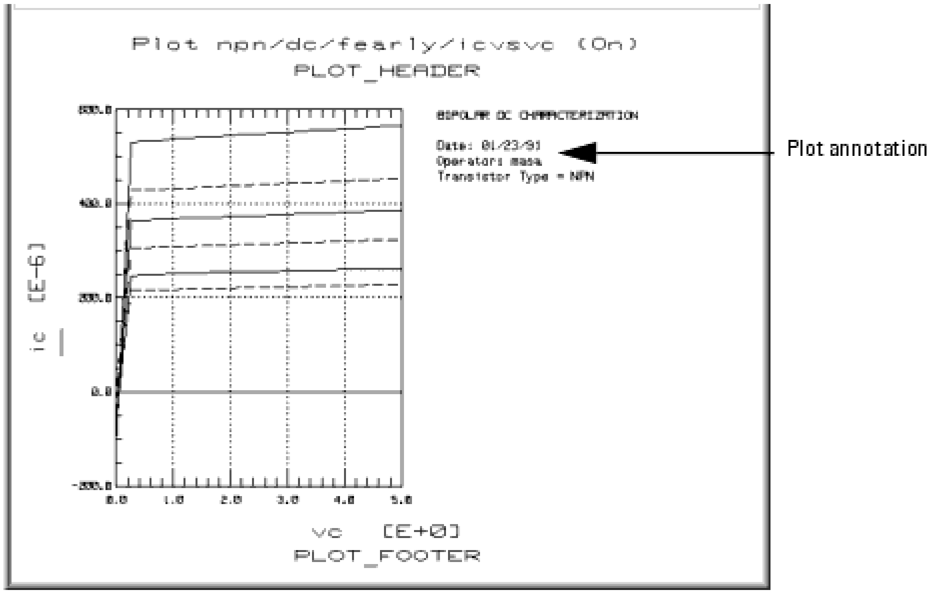
   > **Note**
   > On Windows, you can copy a plot image to the Windows clipboard, paste it into another applications (such as Microsoft Paint), then save it in the other application. To copy a plot image to the Windows clipboard, select **Options > Copy to Clipboard** or press Ctrl_C on the keyboard.

Annotating a Plot Using a PEL Macro

Each plot can read a text file and display this text next to its graph. The text area is 40 characters by 25 lines maximum for the USASCII character set (see *ANNOTATE_CSET* in the section Annotation Variables).

To add annotation text to a plot:

1. Create a Macro program to generate an annotation text.
2. Assign appropriate values to annotation system variables.
3. Open a plot window.



Annotation Variables

The variables that control the annotation in a plot can be defined at any level, providing flexibility and programming capability. These variables are:

- ANNOTATE_MACRO. This variables sets a macro name which is executed by a plot for generating annotation text. If this variable is undefined or blank, then no program is executed. A model name may precede a macro name to locate a macro in another model (for example, /npn/legend).
- ANNOTATE_FILE. This variable sets a file from which a plot reads in annotation text. When this variable is undefined or blank, a plot does not read a file to update its annotation.
- ANNOTATE_AUTO. When this variable is either Y or Yes, then a plot updates its annotation whenever its datasets are updated, for example, when measured or simulated. Otherwise, the annotation is updated when a plot window is opened or its menu choice Update Annotation is selected.
- ANNOTATE_CSET. This is an optional variable to specify a character set for the text. A 16-bit character code is possible by giving a value like jpn, korean, or chinese-t when NLIO (Native Language I/O) is installed. If this variable is undefined or blank, a default character set (usascii as set in $ICCAP_ROOT/config) is used.

Annotation Example

The annotation example shows how to use the annotation text from the example plot used in this section. The following table shows variable values for this example, except ANNOTATE_FILE, which is defined in the *legend* macro program.

System Variables for Annotation

Variable	Value
ANNOTATE_MACRO	legend
ANNOTATE_FILE	
ANNOTATE_AUTO	No
ANNOTATE_CSET	

The *legend* macro program is shown in the following figure. This macro sets the ANNOTATE_FILE variable and writes text into this file. Later, a plot reads this file automatically for its annotation.

A Macro Program for Annotation

```
! Plot annotation generation program
!
file_name = system$("echo $HOME/.icplotnotes")
ANNOTATE_FILE = file_name
x = system("rm " & file_name)
!
printer is file_name
!
print "BIPOLAR DC CHARACTERIZATION"
print
print "Date: "; system$("date +%D")
print "Operator: "; system$("logname")
print "Transistor Type = "; POLARITY
!
printer is CRT
```

Advanced Annotation

Annotation for Each Plot

Multiple plots in a single setup share the same annotation macro and text file. To have different annotation for each plot in a single setup, create a macro program that generates texts and opens each plot in turn. In this case ANNOTATE_MACRO should be blank.

16-bit Code Annotation

If you have installed *NLIO* for 16-bit character code such as Japanese, plots can display these native languages by assigning an appropriate character set to ANNOTATE_CSET. A macro program can include these 16-bit characters in a string and print them out into a file. However, you need to use an external text editor such *vi* in an *hpterm* window to input such characters. To edit a macro program with *vi,* add a new macro program in IC-CAP, write it out to a file, edit this file with *vi,* and then read it back into IC-CAP. IC-CAP can show you these 16-bit characters in a Macro Editor if a 16-bit code font is assigned. For example, for Japanese font, execute the program with the option shown below. Be sure to specify *jpn* for ANNOTATE_CSET in this case.

```
iccap -xrm "iccap*XmText.?*FontList: jpn.8x18"
```

Printing from UNIX

Printing and plotting from IC-CAP on UNIX is accomplished by establishing the desired print setup and then choosing **File > Print**. The Print Setup and related dialog boxes enable you to:

- Select a printer/plotter other than the default
- Install additional printers or plotters
- Set the print resolution (dpi)
- Scale the output

When you select a printer/plotter, you can also change the following default printer/plotter-specific options:

- Page Size
- Source (Paper Tray)
- Duplexing (Double-sided)
- Orientation (Portrait or Landscape)
- Color (Black and White or Color)

When you choose **File > Print**, you can select from the following additional options:

- Choose to send output to a printer/plotter or print to file
- Scale the output to fit to the page
- Select a file format (if printing to file)
- Specify the number of copies

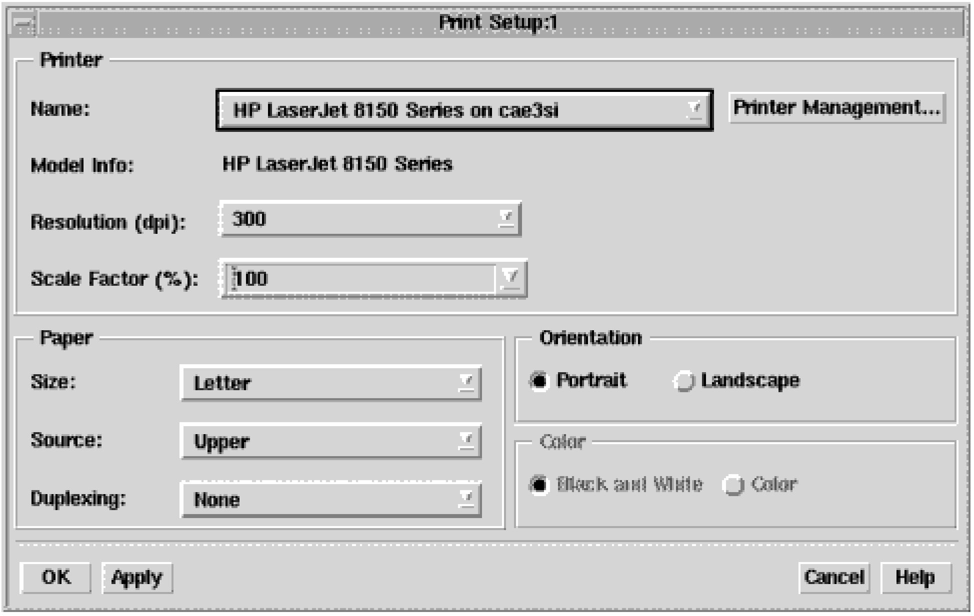
Your print setup is saved in *$HOME/.Xprinterdefaults*. If you do not have a local copy of this file, or the file. *Xpdefaults* (from a previous release), the default file is read from the *$HPEESOF_DIR/xprinter* directory. When you change your print setup, the changes are saved (as new defaults) to *$HOME/.Xprinterdefaults*.

Note
If you do have a file. *Xpdefaults* (from a previous release), the settings of this file are copied to the new filename to serve as the starting point for your print setup. Both files are valid, depending on which release of IC-CAP you are using. The old file is maintained for running an earlier version of IC-CAP, but the new file is used when you run IC-CAP 2006 (or later).

Setting Up a Printer

The *Print Setup* dialog box enables you to setup and manage your printer options. To access the Print Setup dialog box,

1. Choose **File > Print Setup**. The *Print Setup* dialog box appears.



For detailed information on using the Print Setup dialog box, refer to the following table.

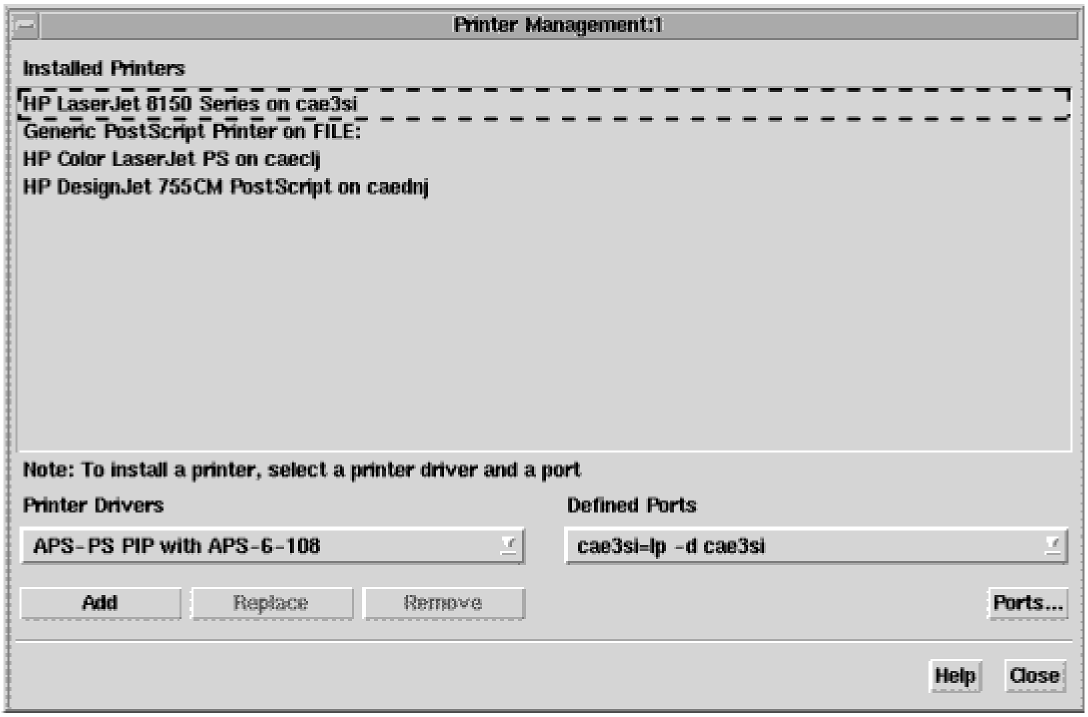
Using the Print Setup Dialog box

Option	**Description**
Printer	Use this section of the Print Setup dialog box to define and manage your printers.
Name:	Select a printer from the Name drop-down list. If the printer you want to use is not available in the list, click the Printer Management button. For more information, refer to Managing Printers.
Printer Management	If you want to add, replace, or remove a printer, click the Printer Management button to access the Printer Management dialog box. For more information, refer to Managing Printers.
Model Info:	This section displays the selected printer model information.
Resolution (dpi):	This option enables you to set the print quality (resolution) in dots per inch (dpi).
Scale Factor (%):	This option enables you to set a scaling factor that defines the percentage of normal size by which to enlarge or reduce the document on the page. Default is 100%.
Paper	Use this section of the Print Setup dialog box to define the paper settings.
Size:	Use the Size drop-down menu to select the paper size. Default is Letter. Note that paper sizes can vary depending on the paper manufacturer. If you own a printer that provides a specific paper size that you are not familiar with, consult your printer manual for information on the paper size options available. The more common American paper sizes are listed below in inches (Width vs. Height). Letter = 8.50" (W) x 11.00" (H) Executive = 7.25" (W) x 10.55" (H) Legal = 8.50" (W) x 14.00" (H) Tabloid = 17.00" (W) x 11.00" (H) Ledger = 11.00" (W) x 17.00" (H) Consult the International Organization for Standardization (ISO) for standard A through D-sizes as well as RA and SRA-sizes.
Source:	Use the Source drop-down menu to select the tray that has the paper you want to use. Default is the Upper tray.
Duplexing:	The Duplexing drop-down menu enables you to print on both sides of the paper. The options available are: - None - No duplexing, the document will only print on one side of the paper. - Flip on Short Edge - This option prints on both sides of each sheet and flips the page along the short edge of the paper. - Flip on Long Edge - This option prints on both sides of each sheet of paper and flips the page along the long edge of the paper. The default Duplexing option is None.
Orientation	Use this section of the Print Setup dialog box to define the orientation of your printed page.
Portrait	Click this option if you want your output printed in portrait orientation mode. Portrait is taller than it is wide when you view the text right-side up. Default is activated.
Landscape	Click this option if you want your output printed in landscape orientation mode. Landscape is wider than it is tall when you view the text right-side up. Default is deactivated.
Color	Use this section of the Print Setup dialog box to define your color settings.
Black and White	Click this option if you want your output in black and white. Default is activated unless the system detects support for color printer. If this is the case, the system will default to the color option.
Color	Click this option if you want your output in color. Default is deactivated unless the system detects support for color printer. If this is the case, the system will default to the color option.

Managing Printers

The *Printer Management* dialog box enables you to manage an individual printer or group of printers. To access the Printer Management dialog box,

1. Choose **File > Print Setup**. The *Print Setup* dialog box appears.
2. Click the **Printer Management** button in the *Print Setup* dialog box. The *Printer Managemen* t dialog box appears.



For detailed information on using the Printer Management dialog box, refer to the following table.

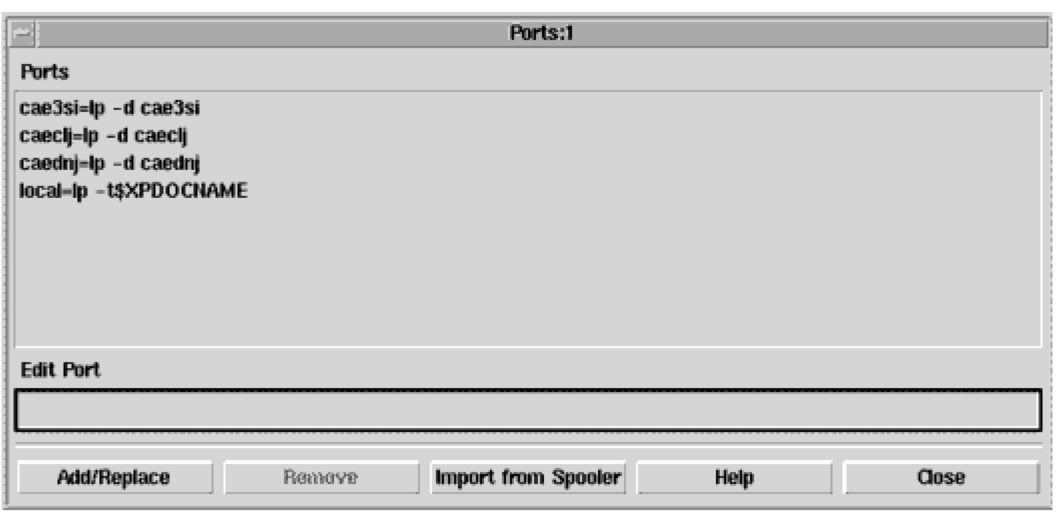
Using the Printer Management Dialog Box

Options	Description
Installed Printers	The Installed Printers field displays a list of all currently installed printers.
Printer Drivers	Use the Printer Drivers drop-down list to select a printer driver.
Defined Ports	Use the Defined Ports drop-down menu to select a defined port.
Ports	To define a new printer port and/or replace or remove an existing printer port, click the Ports button to access the Ports dialog box. For more information on the Ports dialog box, refer to Defining Printer Ports.
Add	To install a new printer, select a printer driver and a defined port, then click the Add button. The new printer appears in the Installed Printers list.
Replace	To replace an existing printer, click the printer you want to replace in the Installed Printers list. Then select a new printer driver and a new defined port. Click the Replace button to replace the printer. The new printer appears in place of the old printer in the Installed Printers list.
Remove	To remove an existing printer or group of printers, click the printer(s) you want to remove in the Installed Printers list and then click the remove button. A confirmation dialog box appears asking if you really want to remove the printer. Click OK to confirm or Cancel to abort the removal process. If you click OK, the printer(s) no longer appears in the Installed Printers list.
Help	Click the Help button to access the online context sensitive help.
Close	Click the Close button to dismiss the Printer Management dialog box and accept the changes. Settings will be saved upon exit.

Defining Printer Ports

The *Ports* dialog box enables you to define new printer ports and/or replace or remove existing printer ports. To access the Ports dialog box,

1. Choose **File > Print Setup**. The *Print Setup* dialog box appears.
2. Click the **Printer Management** button. The *Printer Management* dialog box appears.
3. Click the **Ports** button. The *Ports* dialog box appears.



For detailed information on using the Ports dialog box, refer to the following table.

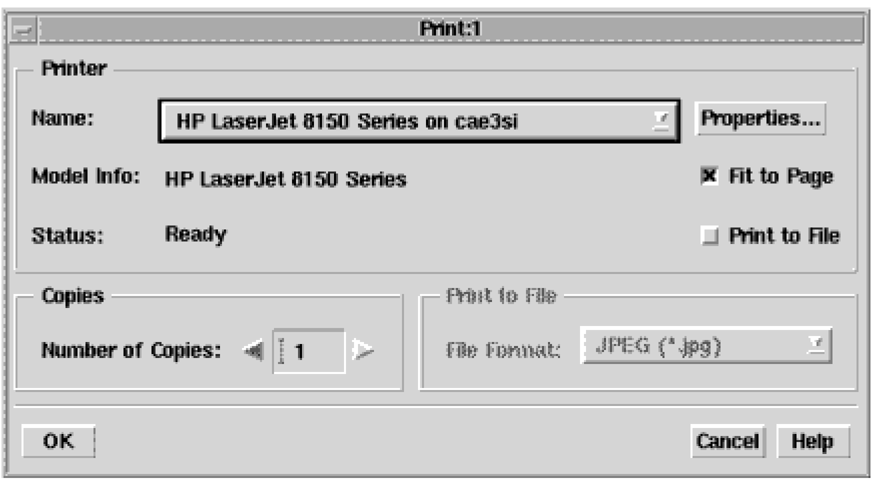
Using the Ports Dialog Box

Option	Description
Ports	The Ports field displays a list of ports. Port names can be any names you choose with the exception of *FILE:* which is a reserved port name.
Edit Port	The Edit Port field is used to enter a new port. After you have entered the port definition, click the Add/Replace button.
Add/Replace	Click the Add/Replace button to update the Ports list with contents of Edit Port field.
Remove	If you want to remove a port from the Ports list, click the port name in the Ports list to activate the Remove button. Click the Remove button to remove the selected port.
Import from Spooler	Click the Import from Spooler button to generate a list of ports (based on your *printcap* file). Note that this third party utility is NOT guaranteed to operate on currently supported platforms.
Help	Click the Help button to access the online context sensitive help.
Close	Click the Close button to dismiss the Ports dialog box and accept the changes. Settings will be saved upon exit.

Printing to a Printer, Plotter, or File

The *Print* dialog box enables you to output your information to a supported printer, plotter, or specified file. To access the Print dialog box,

1. Choose **File > Print**. The *Print* dialog box appears.



For detailed information on using the Print dialog box, refer to the following table.

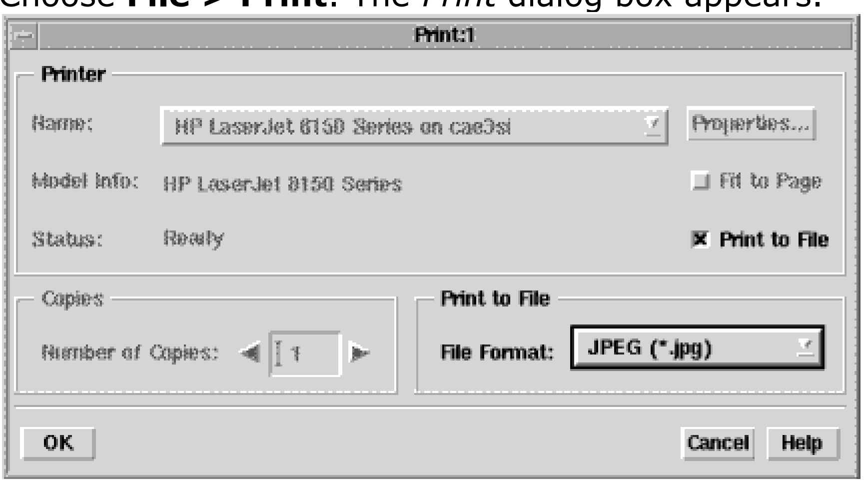
Using the Print Dialog Box

Option	Description
Printer	The Printer options in the Print dialog box enable you to define a printer and the output characteristics.
Name:	Select a printer from the Name drop-down list. If the printer you want to use is not available in the list, click the Cancel button and access the Print Setup dialog box.
Model Info:	This section displays the printer model information.
Status:	This section displays information on the printer status. If the printer is ready to print, the Status will display Ready.
Properties	Click the Properties button to view or set additional options used in the Print Setup dialog box. For more information, refer to Setting Up a Printer.
Fit to Page	Activate the Fit to Page option if you want your output to be automatically scaled to fit on the page.
Print to File	Activate the Fit to Page option if you want your output to be directed to a file. When this option is selected, the Number of Copies feature is deactivated and the Print to File feature is activated. When this option is deactivated, the Number of Copies feature is activated and the Print to File feature is deactivated.
Copies	This section is used to set the number of copies to print.
Number of Copies	Use this feature to specify the number of copies to print. If Print to File is selected, this feature is deactivated.
Print to File	This section is used to define the type of file you want to send your output to.
File Format:	Use this feature to specify the file format to save your output to. After you have selected your file format, click OK in the Print dialog box. A Print to File dialog box will appear enabling you to define the path and name of the file. For more information on the Print to File dialog box, refer to Printing to a File. If the Print to File radio button is deselected, this feature is deactivated.
OK	Click the OK button to accept the settings and send your output to the selected printer or file, depending on how you have set your options. Settings will be saved upon exit.
Cancel	Click the Cancel button to dismiss the Print dialog box. Settings will not be saved upon exit.
Help	Click the Help button to access the online context sensitive help.

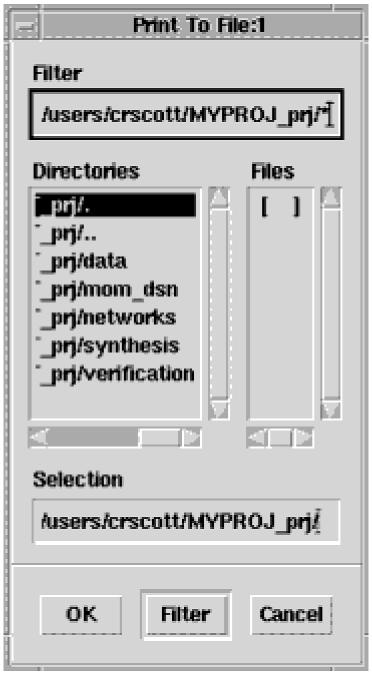
Printing to a File

The *Print to File* dialog box enables you to define a destination path and file name and then output your information to the specified file. To use the Print to File dialog box,

1. Choose **File > Print**. The *Print* dialog box appears.



2. Select the *Print to File* option in the *Printer* section of the Print dialog box. Notice that when you select the *Print to File* option, the *Print to File* section is activated enabling you to select a file format.
3. Use the *File Format* drop-down list to select the desired file format. Options include JPEG, GIF, PDF, Bitmap, and HP-GL/2.
4. Click **OK** in the *Print* dialog box. The *Print to File* dialog box appears.



5. Enter the file name for your output in the Selection field.
6. Click the **OK** button to output the file.

Printing from the PC

This section describes some of the actual printing features available on the PC. For information on basics (such as adding a printer), refer to your Windows documentation.

Printing from IC-CAP on the PC is accomplished by establishing the desired print setup and then choosing **File** > **Print**. Listed below are some of the more common options you can set through the Print Setup and related dialog boxes:

Note
The options available vary based on the printer/printer driver you select.

- Printer (select any installed printer)
- Paper size and source
- Orientation
- Number of copies
- Single- or two-sided printing
- Scaling

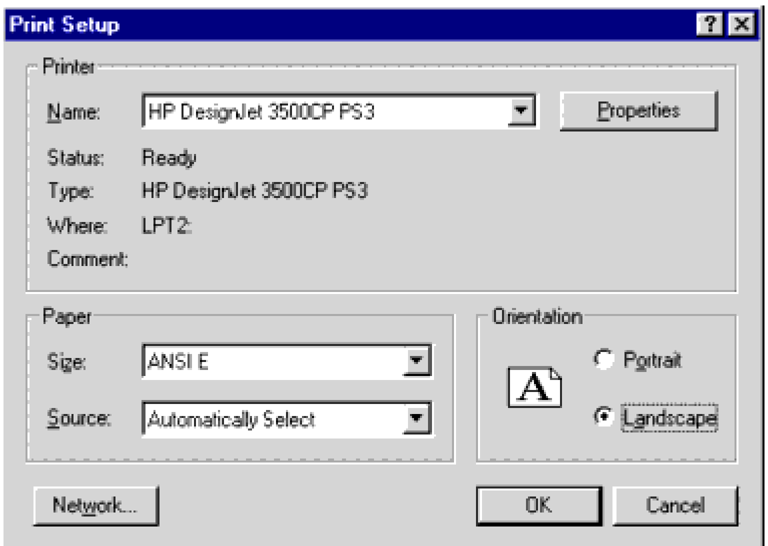
When you choose **File** > **Print**, you can select from the following additional options:

☑ Print to file	Select this option to send output to file for printing at a later time. Select Enhanced Metafile, Windows Metafile, or HP-GL/2 as the file format when the Print to File dialog box appears.
☑ Color output	Select this option to print in color (on a color printer) or in grayscale, rather than black and white (on a monochrome printer).
☑ Copy to clipboard	Select this option to place the image on the Windows clipboard (Bitmap) for pasting in any Windows application.

Establishing a Print Setup

To establish a print setup:

1. Choose **File** > **Print Setup**.
2. Select the desired printer from the drop-down list.



3. Change any of the options here as desired, or click **Properties** to set additional options, such as *Scaling*. Note that the appearance of the Properties dialog box varies depending on the selected printer.
4. Change any other options as desired and click **OK** to dismiss the Properties dialog box.
5. Click **OK** in the Print Setup dialog box and you are ready to print.

Basic Printing

To send the entire contents of the window to the printer:

1. Choose **File** > **Print** and a dialog box appears.
2. Change any print options as needed, and click **OK**.

Automating Print Functionality

The following print functionality can be automated via PEL:

- Print to the following supported file types:
  - hpgl2
  - black and white postscript
  - emf for the PC only
- Display the File > Print dialog box, which enables the user to select a printer and to request a printout.

Starting with IC-CAP 2004, you can no longer send a print file to the printer from PEL without user intervention. You can not create print files other than the ones listed above. For example, you can not create a color postscript file formatted for a color printer. To generate that type of file, you must use the File > Print dialog boxes.

Also, the semantics of the Print Via Server command have changed slightly with IC-CAP 2004. In previous releases, if a printer was setup as the default, issuing a Print Via Server command would print the file with no user intervention. Now, the Print Via Server command *always* displays the print dialog box requiring user intervention. See the following examples.

```
 ! Displays dialog box as though File > Print menu had been
! selected. You cannot anticipate the dialog with
! optional arguments, it will always come up.
iccap_func\("myPlot","Print Via Server"\)
! following line creates an HPGL2 file
iccap_func\("myPlot","Dump Via Server","Y","hpglfile.hgl", "HPGL2"\)
! following line creates a black & white postscript file
iccap_func\("myPlot","Dump Via Server","Y","psfile.ps","PS"\)
! following line creates a .emf file on PC only
iccap_func\("myPlot","Dump Via Server","Y","emffile.emf", "EMF"\)
! Note, the following form of the command worked in
! previous releases of IC\-CAP but is no longer supported.
iccap_func\("myPlot","Dump Via Server","N"\)
! Displays dialog box as though the Print button had been
! pressed on a tabular data window
iccap_func\("myOutput","Print Via Server"\)
```

For additional information, see *Dump Via Server* (prog) and *Print Via Server* (prog).

Macros

The following topics are covered under Macros:

- Introduction and Basics
- Creating and Running a Macro
- Saving Macros
- Macro Features
- Adding a Macro to the Model

Introduction and Basics

This section contains procedures for creating, editing, and using macros. Macros can provide a powerful tool for automating routine or repetitive procedures by increasing the efficiency of your operations. Macros, consisting of short programs or scripts that execute IC-CAP commands in a predetermined sequence, automate routine test procedures so that the execution of a single command measures, extracts, simulates, optimizes, plots, and reports on the characteristics of the tested device or circuit.

Macros are simply a way to automate routine operations or to simplify complex operations. For example, you can create a macro to do the following:

- Measure all four setups of the *dc* DUT in the *npn* model.
- Run extractions, simulation and optimization.
- Display all plots in a model.

In addition to these routine tasks, you can perform more sophisticated operations, such as:

- Writing data to an external data base.
- Making choices and branches, based on the value of data sets, parameters or variables.

Pre-defined macros are provided for each model and you can execute macros manually or automatically upon IC-CAP startup. Macros are easy to write-the facilities for creating, editing, and running macros are available in the Macros folder. Each model can have any number of macros, as the example in the next section demonstrates.

Creating and Running Macros

Refer *Creating and Running a Macro* (prog) for further details.

Saving Macros

Refer *Saving Macros* (prog)

Macro Features

Refer *Macro Features* (prog)

Adding a Macro to the Model

Refer *Adding a Macro to the Model* (intro)

Transforms

Creating and Running a Transform

A transform is a framework for mathematical or logical functions that operate on data. Extraction and optimization are accomplished through the use of transforms. A transform can operate on any combination of data sets, parameters, and variables. These inputs are used to calculate either a new data set or new values for parameters and variables. Transforms are created for Setups, since they typically operate on data from a particular Setup.

To create and run a transform:

1. Open the Model window and select the desired Setup.
2. Click the **Extract/Optimize** tab and note the list of existing transforms.
3. Click **New** and provide a name for the new transform in the dialog box that appears.
4. Click **OK** and the new transform name is added to the list.
5. Select an existing function from the Browser, or type `Program2` or `Program` in the Function field, and press Enter to create your own.
6. Fill in the argument fields displayed for a selected function, or type the desired text if creating your own.
7. To run the transform, click **Execute**.

Related Topics:

Using Transforms and Functions (prog)
Creating and Running Macros (prog)
Parameter Extraction Language (prog)

Creating a Function

To create a new function:

1. Click the **Extract/Optimize** tab.
2. Select the transform for which you want to create a function.
3. Type `Program2` or `Program` in the Function field and press Enter. The window changes to display a scrollable text field.
4. Type the desired text.

	Note When you create a function within the user interface, you use a BASIC-like language referred to as Parameter Extraction Language (PEL). You can also create functions outside IC-CAP using C language.

Selecting an Existing Function

To choose a function from the Browser:

1. Select the transform for which you want to assign a function and click the **Browse**

button.
2. In the dialog box that appears, select the appropriate Function Group to display a list of available functions in that group.
3. Select a Function and a brief description is displayed.
4. Click **Select** and the dialog box disappears and the function name appears in the Function field.
5. Where applicable, assign appropriate values or edit as needed.

Related Topics:

Using Transforms and Functions (prog)
Creating and Running Macros (prog)
Parameter Extraction Language (prog)

Saving Transforms

By default, transforms are saved when you save the model file. But you can explicitly save a transform at any time and specify another directory, if desired.

To save transforms:

1. With any of the transforms in the Extract/Optimize folder selected, choose **File > Save As**.
2. Select the **Transform** option.
3. Select the transform name from the drop-down list.
4. If desired, change the path using the Browser.
5. If desired, supply a different filename in the File Name field.
6. Click **OK**.

Related Topics:

Using Transforms and Functions (prog)
Creating and Running Macros (prog)
Parameter Extraction Language (prog)

File and Data Management

The directory containing the example model files is protected to preserve an original copy of each model file. The typical method of modifying a model file is to use the Save As command and save a copy of it to another directory. You will probably want to create one or more directories, in any terminal window, for storing your model files and data.

Quick access to the example model files is provided with the File > Examples command in the Main window. When you want to open a model file other than one from the examples directory, by default, the path in the dialog box is set to the directory from which you started the program. Double-click to get to the desired directory.

> **Note**
> A symbol displayed in the Main window represents a model file currently in memory. If you close a Model window using File > Close, the file remains in memory and can be reopened by double-clicking the symbol.

Opening Files

To open a file:

1. Choose **File > Open** and a dialog box appears. By default, the path in the Filter field is set to the directory from which you started the program and the filter is set to *.ext where ext is an extension specific to the task you are performing. (For example, *.mdl for model files, *.hdw for hardware configuration files, etc.). All files in the current directory with that extension are displayed.

2. Adjust the path as needed. You can type in the Filter field and then click the **Filter** button or use the mouse to navigate the list of Directories.

   > **Note**
   > The path for IC-CAP model files cannot contain any folder names that use a space. For example, *C:\Model Files\IC-CAP 2004*. If a model file is saved in a folder name with spaces, you will not be able to open the model file. You will have to move the model file to a folder name that does not use a space.

3. To choose a file from the Files list box, double-click it or click once and choose **OK**. (Tip: If you click once, you will see the selected filename reflected in the Selection field.)

   > **Note**
   > If loading a model file, once it is loaded, a symbol with the model name is displayed in the work area. Double-click the symbol to open the Model window.

Opening Parts of a Model File

The **File > Open** command in the Model window enables you to open a file (previously saved using File > Save As in the Model window) that contains some portion of a model. To open a file containing a portion of a model:

1. If you want to replace some portion of the current model file with a portion of another file, select the item you want to replace.
2. Choose **File > Open** in the Model window.
3. Select the appropriate file type and click **Browse** to view the files of that type in the directory you specify.

	Note The path for IC-CAP model files cannot contain any folder names that use a space. For example, *C:\Model Files\IC-CAP 2004*. If a model file is saved in a folder name with spaces, you will not be able to load the model file. You will have to move the model file to a folder name that does not use a space.

4. Select a file and click **OK**.
5. For Model Parameters and DUT Parameters, select **Replace Parameter Set** or **Read Values Only** and click **OK**.
   - Replace Parameter Set replaces all Value, Min, and Max values with the ones in the selected file. If the selected file does not include Min and Max ranges (e.g., when reading pre-IC-CAP 2004 parameter set files), the existing Min and Max ranges are deleted.
   - Read Values Only replaces parameter Values while maintaining existing Min and Max ranges if possible. If the new Value is outside the existing range, Min or Max is extended to include the new Value and a warning is displayed in the Status window.
6. For all other Model folders, select the **Replace** option to use the specified file as a replacement and click **OK**. To add the specified file, make sure the option is turned off and click **OK**.

Saving Files

To specify a directory and a filename:

1. Choose **File > Save As** and a dialog box appears.
2. Double-click in the Directories list box to locate the desired directory.
3. To save to an existing file, select that filename from the Files list box and click **OK**. To save to another name, type that name in the Selection field and click **OK**.
4. Where applicable, click **OK** to dismiss the previous dialog box.

Saving a Model File

From the Save As dialog box in the Main window:

1. Select all models you want to save to one model file.
2. Accept the default status of saving the file with the measured and simulated data, or enable the option to save without it.
3. Adjust the path and filename as needed, and click **Apply**.

Hint: To avoid typing a lengthy path change, use the Browser.

> **Note**
> The path for IC-CAP model files cannot contain any folder names that use a space. For example, do not save to *C:\Model Files\IC-CAP 2004*. If you include a space in a folder name, you will not be able to load the model file. You will have to move the model file to a folder name that does not use a space.

4. Continue saving as needed, clicking **Apply** to effect each change.
5. Click **OK** to dismiss the dialog box.

> **Note**
> You can also save a model file from the Model Window if you want to save all components to one file.

Saving Parts of a Model File

When you choose **File > Save As** in a Model window, you can choose which part of the model to save to file: DUT, setup, input, output, transform, plot, macro, variable table, parameter set, circuit description, or instrument options.

1. Select the option representing the portion of the file you want to save. (Where applicable, select the option from the drop-down list.)
2. Accept the default status of saving the file with the measured and simulated data, or enable the option to save without it.
3. Adjust the path and filename as needed.

Hint: To avoid typing a lengthy path change, use the Browser.

> **Note**
> The path for IC-CAP model files cannot contain any folder names that use a space. For example, do not save to *C:\Model Files\IC-CAP 2004*. If you include a space in a folder name, you will not be able to load the model file. You will have to move the model file to a folder name that does not use a space.

4. Click **OK** to effect the save and dismiss the dialog box.

Changing Default Directory

By default, model files and data will be saved to the directory from which you started the program.
To change the directory:

1. Choose **File > Change Directory**. By default, the path in the Filter field is set to the directory from which you started the program.
2. Adjust the path as needed. You can type in the Filter field and then click the **Filter**

button or use the mouse to navigate the list of Directories.

3. When the Selection field reflects the desired directory, click **OK**.

Importing Data into a Single Setup

1. Select the DUT and setup.
2. Select **File > Import Data** and choose **Active Setup**.

    Alternatively, you can select **Measure/Simulate** and click **Import Data**.
3. Enter the name of the file directly into the field corresponding to the setup, by typing in or by using the File Browser.
4. Select a fill data type.
    - **Measured if available, otherwise Simulated:** Outputs of type M or B receive mdm data in their measured array. Outputs of type S receive mdm data in their simulated array.
    - **Measured only:** Outputs of type M or B receive data in their measured array. Outputs of type S do not receive any data.
    - **Simulated if available, otherwise Measured:** Outputs of type S or B receive mdm data in their simulated array. Outputs of type M receive mdm data in their measured array.
    - **Simulated only:** Outputs of type S or B receive mdm data in their simulated array. Outputs of type M do not receive any data.
5. Click **OK**. The data is now imported into the outputs of the setup.

Importing Data into All Setups in an Active DUT or All DUTs in the Model

1. Select the DUT.
2. Select **File > Import Data** and choose one of the following:

    All Setups in Active DUT
    All DUTs in Model
    The dialog box that appears has three columns (DUT, Setup, Data Filename) and a Fill Data Type selection area.
3. Specify a filename for each setup. You can type the path and filename directly in the Filename field, or use the Browser to select it.
4. Click **Apply** each time you specify a filename for a setup.
5. Select a fill data type.
    - **Measured if available, otherwise Simulated:** Outputs of type M or B receive mdm data in their measured array. Outputs of type S receive mdm data in their simulated array.
    - **Measured only:** Outputs of type M or B receive data in their measured array. Outputs of type S do not receive any data.
    - **Simulated if available, otherwise Measured:** Outputs of type S or B receive

mdm data in their simulated array. Outputs of type M receive mdm data in their measured array.
- **Simulated only:** Outputs of type S or B receive mdm data in their simulated array. Outputs of type M do not receive any data.

6. Click **OK**.

Tips

- To view the header of an MDM data file, select the Setup cell in the dialog box, enter the corresponding MDM filename, then click **View**.
- To create the inputs and outputs for a new setup from the header of an MDM file, first create a new setup, select the setup name in the dialog box, enter the corresponding MDM filename, and click **Create**.

See Also

- Importing Data into a Single Setup
- *MDM Data* (data)