

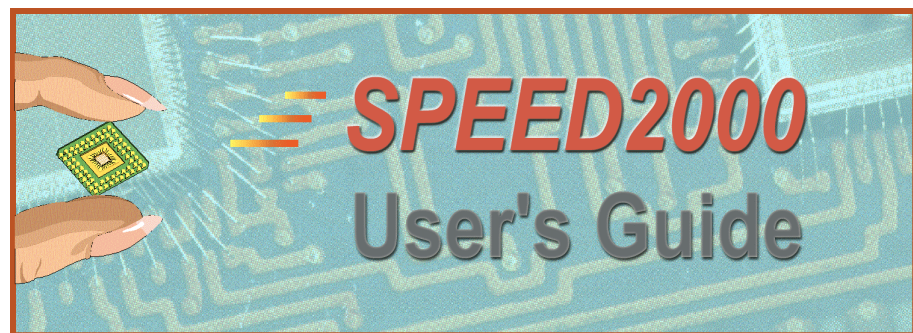


# ***SPEED2000***

**S**IMULATION **P**ACKAGE FOR **E**LECTRICAL **E**VALUATION AND **D**ESIGN

---

**A New Generation of Power and Signal Integrity Tool**



SIGRITY

# **SPEED2000™ User's Guide**

---

© Copyright SIGRITY, Inc. 1997-2001  
4675 Stevens Creek Blvd. · Suite 130  
Santa Clara CA 95051  
Phone 408.260.9344 · Fax 408.260.9342  
<http://www.sigrity.com>

Last update: June 27, 2002

---

---

# **SPEED2000 User's Guide**

## **Table of Contents**

---

### **Welcome !**

What Is SPEED2000? .....	XI
File Format Translators .....	XI
What Is SPEED2000 Used For? .....	XII
Who Should Use SPEED2000? .....	XII
Prerequisites .....	XII
Overview of SPEED2000 Use .....	XIII
About This Manual .....	XV
Best Wishes To You ! .....	XV

### **1 Getting Started.....1-1**

SPEED2000 — A Simple Example .....	1
Inside the Example .....	2
Running the Example SPEED File -- Using the Simulator .....	3
Some Basic Techniques For Using SPDGEN .....	9
Building a Simple SPEED2000 File — Using the Generator .....	23
Congratulations! .....	37

## **2 SPDGEN Features Overview .....2-1**

Menu Option Citation .....	1
Initial/Main Menu Bar.....	2
File Menu (before package loaded) .....	2
View Menu (before package loaded).....	3
Help Menu .....	5
Toolbar - Main (before a package is loaded).....	5
Status Bar .....	6
SPDGEN Workspace: Layer View or 3D View .....	6
The Layer View Window .....	7
ICON.....	8
File Menu (after package loaded) .....	9
Edit Menu .....	11
View Menu (Layer View).....	14
Setup Menu .....	16
Background Menu .....	18
Tools Menu .....	19
Window Menu .....	23
Help Menu .....	23
The 3D View Window .....	24
View Menu (3D View) .....	25
The Toolbars of SPDGEN.....	26
Toolbar - Main.....	26
Toolbar - Object.....	26
Toolbar - Checking .....	27
Toolbar - Select.....	27
Toolbar - Circuit .....	28
Toolbar - Shape.....	28
Toolbar - 3D .....	29

## **3 SPDSIM Features Overview .....3-1**

Menu Option Citation .....	1
Initial/Main Menu Bar.....	2
File Menu (before package loaded) .....	2
View Menu .....	4
Help Menu .....	6
Toolbar - Main (before a package is loaded).....	6
Status Bar .....	6
The Package View Windows .....	7
Error Log File .....	8

Common Window Menus.....	8
File Menu (after package loaded).....	8
Help Menu .....	11
Window Menu .....	12
Toolbar - Main (after the package is loaded) .....	12
2D Curves Window .....	13
View Menu (2D Curves window) .....	15
Tools Menu.....	17
2D Curves Toolbar .....	23
Right-click Pop-Up Menus (context sensitive) .....	23
Right-click Present Curves Menu.....	24
Right-click Loaded Curves Menu.....	27
Right-click Plotted Information Menu (Time Domain) .....	29
Right-click Menus (Frequency Domain) .....	33
Spatial Distribution Window .....	33
View Menu .....	34
Spatial Distribution Toolbar .....	36
Right-click Pop-Up Menus (context sensitive) .....	37
Layer Display Window .....	39
View Menu .....	40
Analysis Menu.....	41
Layer Display Toolbar.....	41
Package Display Window .....	42
View Menu .....	42
Package Display Toolbar.....	43

## **4 Working with the Package.....4-1**

Opening, Closing and Saving Files.....	1
File Saving Options Dialog Box .....	2
Auto Save .....	3
Using Zoom Functions.....	3
Viewing the Structure in 3D .....	4
Viewing Layers and the Layer Selection Dialog Box .....	5
Display of Shape Modes .....	6
Setting Up the (Layer) Stack-up .....	7
Net Management.....	15
“Unnamed” Nets.....	15
Showing Objects .....	20
Selecting Objects .....	21
Setting Up the Display, Drawing and Mesh Environment .....	22
Using Drawing Tools.....	24

Using Node Tools.....	25
Working on Shapes .....	29
Positive and Negative Shape Objects .....	29
Holes and Voids .....	30
Shape (and trace) Process function.....	30
Polygon Smoothing/Simplifying .....	31
Working with Shape Definitions .....	32
Drawing Boxes .....	43
Drawing Circles.....	44
Drawing Polygons .....	45
Deleting Shape Objects .....	46
Viewing and Editing Shape Properties.....	49
Moving and Editing Shape Components.....	50
Pre-Conditioning the .spd File For Simulation .....	55
Polygon Smoothing .....	62
Special Voids — Dog-Leg Holes and Small Holes.....	63
Working on Vias .....	68
Via Radius Map.....	68
Via Representation On the Layer View Window .....	75
Working on Traces .....	91
Working on Coupled Lines .....	95
Traces Over “Split Planes”.....	97
Checking Drawing Errors .....	99
Error Log File.....	102
Modifying Object Properties.....	103
Merging Two .spd Files .....	103
Moving and Rotating a Package.....	105
Loading DXF Files as Background.....	109
Loading a DXF File for the First Time .....	109
Loading DXF Files Using Saved Settings of Dimension and Position.....	111
Changing Scale and Unit before Loading a DXF File .....	112
Finding Objects .....	113
Object Names and Wildcards.....	113

## **5 Working with Circuits .....5-1**

Overview Of The Linkage Dialog Box .....	2
Adding New Partial Circuits .....	5
Creating New Partial Circuit Definitions.....	5
Loading an External Partial Circuit Definition File.....	6
Deleting a Partial Circuit.....	7
Deleting a Partial Circuit Definition .....	7

Editing a Partial Circuit Definition .....	8
Linking Partial Circuits to the Package .....	9
Deleting a Linkage .....	14
Modifying a Linkage .....	14
Checking Linkages .....	14
Selecting Circuits For Simulation .....	15

## **6 Establishing Simulation Parameters .....6-1**

Establishing Computation Parameters .....	2
Establishing the numerical mesh .....	2
The Mesh Dialog Box: .....	2
To turn on the mesh for viewing: .....	3
Error Checking with the Mesh On .....	4
Establishing Transient Simulation Parameters .....	5
The Transient Dialog Box .....	5
Specifying Simulation Time .....	5
Enabling Metal Plane Skin Effect Option .....	5
Enabling Transmission Line Metal Loss Option .....	6
Enabling Dielectric Loss and Dispersion Option .....	6
Enabling Initial DC Analysis Option .....	6
Passivity Option .....	7
Edge Boundary Condition .....	7
Choosing the Viewing Parameters .....	8
Choosing Circuit Voltages for Viewing .....	9
The View Circuit Voltage Dialog Box: .....	10
Choosing Package Voltages for Viewing .....	11
The View Package Voltage Dialog Box: .....	11
Choosing Circuit/Package Currents for Viewing .....	13
The View Current Dialog Box (Circuit): .....	13
The View Current Dialog Box (Trace): .....	14
The View Current Dialog Box (Via): .....	15
Choosing 3D Spatial Voltages for Viewing .....	16
The View 3D Dialog Box: .....	17
Choosing Parameters for Voltage Viewing and Analysis .....	18
The Distribution Dialog Box: .....	18
Far Field (Edge) Radiation .....	19

## **7 Simulation and Post Processing .....7-1**

SPEED2000 Data File Flow .....	1
Starting Simulation .....	3

Controlling Simulation.....	5
The Spatial Distribution Window .....	5
To view the spatial distribution of voltage between adjacent layer(s):	6
To view the voltage value at a selected location: .....	7
The 2D Curves Window .....	8
To view circuit/package voltages: .....	8
Viewing Transient Circuit/Package Currents.....	9
To view circuit/package currents:.....	9
To export the curve display window as a bitmap file: .....	9
The Layer Display Window .....	10
The Package Display Window .....	10
Post Processing .....	10
Viewing Probability Distribution of Power/Ground Plane Noise .....	10
To view the probability distribution function of the noise voltage: ..	11
Generating Fourier Transforms of Transient Data .....	12
To view voltages in the frequency domain: .....	12
To view currents in the frequency domain: .....	12
Obtaining Frequency Domain Parameters.....	13
Using the Fourier Calculator (an example) .....	13
Saving Curve Data into Data Files .....	18
To change the name of a curve: .....	18
Loading and Plotting Previously Saved Curves .....	18
To load saved curves from data files: .....	18
Printing the Results.....	18

## **8 Calculating Radiation ..... 8-1**

Selecting Traces and Edges for Radiation Calculation .....	1
Launch Radiation Calculation in SPDSIM .....	3
Trace Tab.....	3
Edge Tab .....	4
FCC Tab .....	5
2D Pattern Tab .....	9
Printing the Results .....	10

## **9 Batch Mode Processing ..... A-1**

SPDSIM .....	2
SPDGEN .....	3

## **Index.....1**





## Welcome !

Welcome to SPEED2000! You are about to begin “Achieving What Others Can’t” using a new type of power and signal integrity software tool.

## What Is SPEED2000?

SPEED2000 is a new generation of power and signal integrity tool for electrical simulation of electronic packages. It integrates circuit and transmission line simulations with a fast, special-purpose electromagnetic field solver that computes electromagnetic interactions in multi-layer chip packages, multi-chip modules (MCMs) and printed circuit boards.

In SPEED2000, power and ground planes are not treated as ideal power and ground planes with fixed potentials; rather, electromagnetic interactions inside of packages (such as package resonance, coupling between different components, as well as interactions between circuits and packages) are taken into account during simulations. Transient as well as spatial variations of voltages and currents on power and ground planes are all computed and can be easily visualized. The signal waveforms, computed by SPEED2000, automatically contain the effects of power and ground voltage bounces.

### ***File Format Translators***

The SPEED2000 family of products includes file format translators for BRD, NDD, PADs and DSN file formats. These translators make it possible to integrate the functions and capabilities of SPEED2000 to your existing design processes and tools.

Very important — your purchase of SPEED2000 includes a committed technical support and documentation team. In many cases, our dedicated support engineers will personally address your questions. And finally, please visit our website: [www.sigrity.com](http://www.sigrity.com) for additional information about product updates and training.

## What Is SPEED2000 Used For?

SPEED2000 is a versatile electrical simulation tool for the analysis and design of electronic packages of various kinds, including single chip carriers, multi-chip modules and printed circuit boards.

SPEED2000 is particularly effective to be used for —but it is not limited to— the following applications:

- Computation of power and ground noise.
- Evaluation and design of power and ground distribution systems, including power and ground plane arrangement, and power and ground via/pin assignment.
- Determination of decoupling capacitor placement, including the number, values, and locations of decoupling capacitors.
- Identification of package resonance.
- Evaluation of electromagnetic coupling between different components.
- Evaluation of signal and noise waveforms at various locations.
- Evaluation of signal and noise spectrum at various locations.
- Determination of frequency-dependent port parameters of packages, such as port input impedances, S parameters, and transfer functions between different ports.
- Evaluation of electromagnetic radiation from packages and printed circuit boards.

## Who Should Use SPEED2000?

SPEED2000 is intended to be used by engineers and researchers who work in the areas of design, modeling, and simulation of integrated circuits, chip carriers, multi-chip modules and printed circuit boards.

### ***Prerequisites***

Users should have at least an entry-level experience in SPICE type of circuit simulation, and be aware of “signal integrity” issues in high-speed or high-frequency electronic systems.

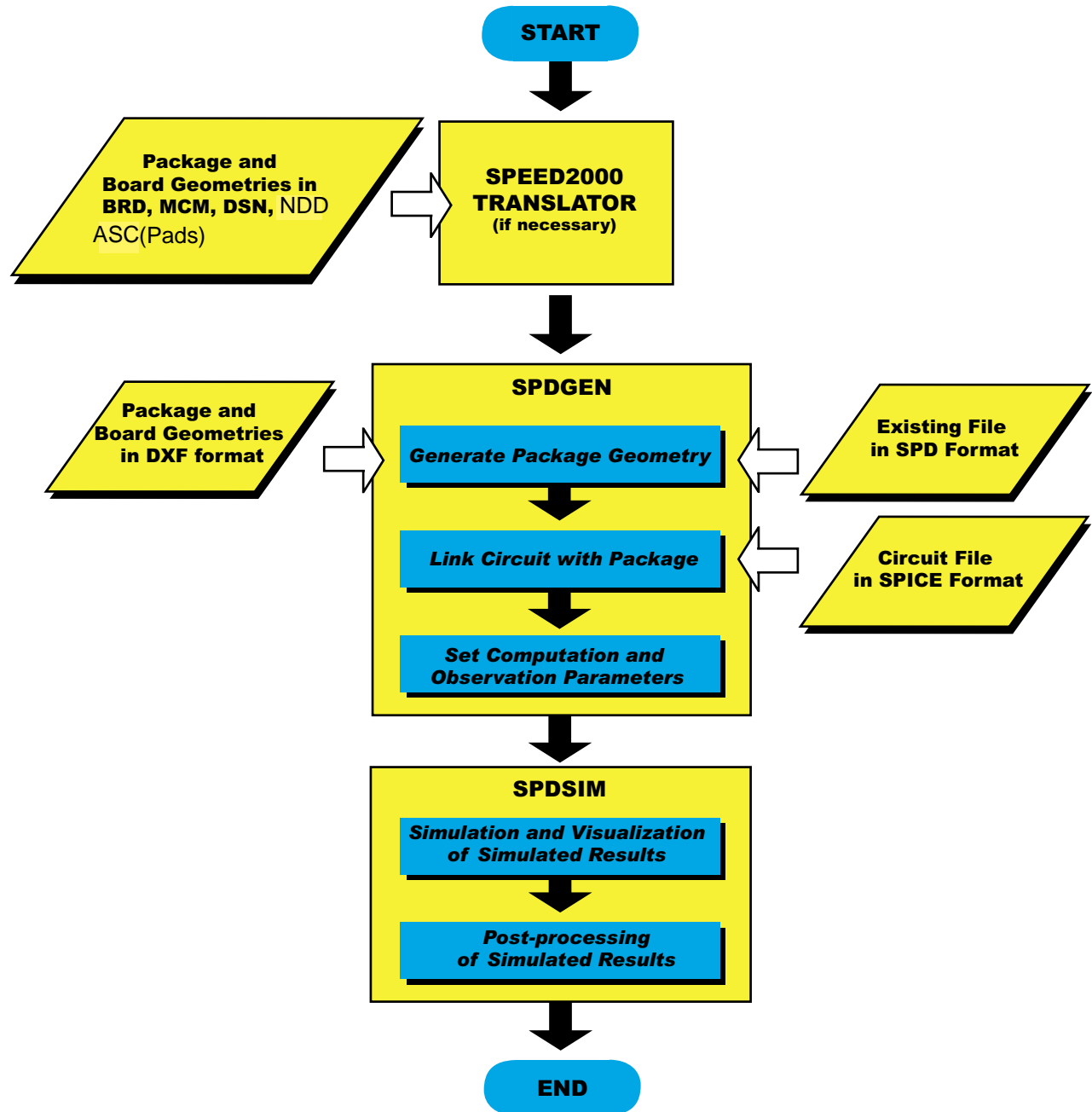
## Overview of SPEED2000 Use

SPEED2000 primarily consists of two executable programs: SPDGEN and SPDSIM. SPDGEN is used for SPEED2000 (.spd) file building and SPDSIM is used for simulation and post-processing. Several translators are also provided to complement SPEED2000. These translators enable the user to conveniently transform the file formats generated by other EDA tools to SPEED2000 (.spd) file format. The SPEED2000 family of products includes file format translators for BRD, MCM, NDD, PADs and DSN file formats.

An overview of SPEED2000 operation is shown in Figure i. As shown by the figure, SPDGEN is used for generating package geometry, importing circuit files, specifying the linkages between circuits and the package components, and setting computation and visualization parameters. Circuit files imported to SPDGEN are in SPICE format. Previously generated physical package layouts can be translated from other EDA programs. These translated files are then read into SPDGEN. SPDGEN can also read files (describing the package geometry) in DXF format.

The SPDSIM module performs the transient waveform simulation. With SPDSIM users can visualize the spatial voltage variations on power and ground planes, and/or transient voltages and currents in circuits and packages. The post-processing capabilities of SPDSIM allow you to:

- Transform the simulation results from time-domain to frequency domain.
- Obtain frequency-dependent characteristics such as port input impedances and transfer functions.
- Save simulated results to data files.
- Load previously saved data.
- Print out simulated results
- Display electromagnetic radiation levels from traces and from edges of metal planes.



**Figure i. Work Flow for SPEED2000**

## About This Manual

This SPEED2000 User's Guide introduces all the functions of SPEED2000. After installation, the best way to get started is to try the example provided in the section "Running a Simple Example" (in Chapter 1). SPEED2000 is composed of two executable modules: SPDGEN and SPDSIM. Chapter 2 presents a features overview of the SPDGEN module. Chapter 3 presents a features overview of the SPDSIM module. Chapter 4 discusses all aspects of working with the package. Chapter 5 discusses working with the circuits and linking circuit nodes to the package nodes. Chapter 6 discusses the simulation parameters. Chapter 7 details the user interface of the SPDSIM simulator module, and details post processing capabilities. Chapter 8 details how to use the Radiation functions to calculate radiation results. Appendix A currently contains a re-direct message. Appendix A discusses batch mode processing. The Index of this document is designed to be a hypertext-tool to help you quickly locate information.

The details of the .spd file format are available in a separate document — see [.spd File Format Reference](#) document. User information for the various translators is available in the [Translators](#) document. Both documents are available on the Sigrity website [www.sigrity.com](http://www.sigrity.com)

This User's Guide is designed to be useful in either print or OnLine form. The Online form, contains numerous hyperlinks for navigation and it is kept up-to-date, with the latest information from Speed2000 developers. This document is also available in it latest and downloadable (PDF) form on our website: [www.sigrity.com](http://www.sigrity.com)

## Best Wishes To You !

We believe our SPEED2000 tool will significantly help you in designing new IC packages and printed circuit boards with superior electrical performance. SPEED2000 will also assist you in identifying and removing defects or improper designs in your existing systems — and finally, SPEED2000 will help you reduce the time-to-market cycle of your products.

If you encounter any difficulties in using SPEED2000, or if you have any suggestions for improvements, please let us know.

- Contact us by telephone at: 408.260-9344
- mail us: [info@sigrity.com](mailto:info@sigrity.com) or [support@sigrity.com](mailto:support@sigrity.com)
- Our website address is: [www.sigrity.com](http://www.sigrity.com). You will find the very latest edition of the User's Guide, new product information and product updates posted here.

We will try our best to make your use of our product effective and your work successful.

## Getting Started

This chapter contains information to get you started using SPEED2000. The details of installation and simple step-by-step examples are provided. Subsequent chapters provide a detailed treatment of the processes and procedures we introduce in this chapter.

### SPEED2000 — A Simple Example

To get a general feeling of how to use SPEED2000, **read this section carefully and follow the instructions step by step.** SPEED2000 is comprised of two executable modules: SPDSIM and SPDGEN. You will learn how to use the simulator module, SPDSIM, to observe the simulation results. You will also use the generator module, SPDGEN, to create a file, which contains your package and circuit information in a file format (.spd) that SPEED2000 products understand.

**NOTE:**

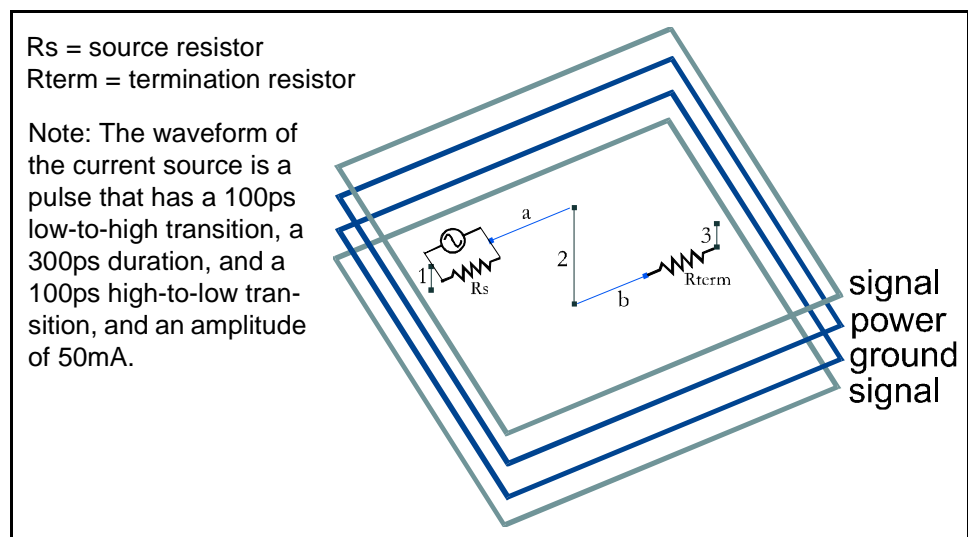
Later chapters will go into great detail about all aspects of SPEED2000; our intent here is to just get you started using SPEED2000.

The SPD file “EXAMPLE.spd” that comes with the installation is a simple, but complete, pre-built SPEED2000 format file (.spd). In the rest of this chapter, we will:

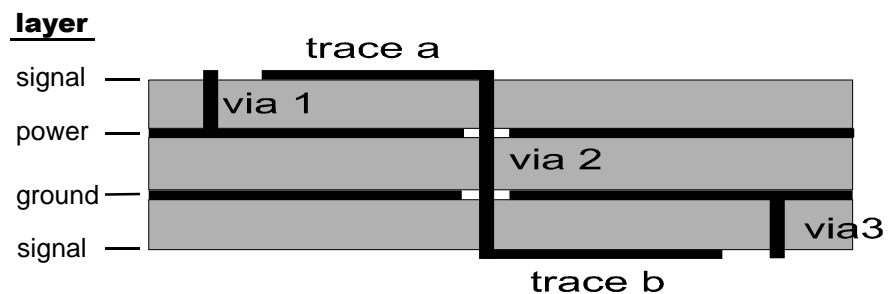
- Explain the example file, EXAMPLE.spd.
- Demonstrate how to load the .spd file into the SPDSIM module,
- Perform a simulation using SPDSIM.
- Perform post-processing using SPDSIM.
- Demonstrate how to create a .spd file, like EXAMPLE.spd, using the SPDGEN module.
- Demonstrate some basic techniques for using SPDGEN:
  - [“Start SPDGEN” on page 1-9,](#)
  - [“Open a New .spd File” on page 1-10,](#)
  - [“Setting Up Layers” on page 1-11,](#)

- [“Activate a Layer and Viewing Layers” on page 1-12,](#)
- [“Close the Current .spd File” on page 1-13](#)
- [“Drawing Traces —also Select/View/Move/Delete Objects, Undo/Redo\)” on page 1-14,](#)
- [“Adding Vias” on page 1-15,](#)
- [“Check Drawing Errors” on page 1-16,](#)
- [“Connect Traces/Merge Nodes” on page 1-19](#)
- and [“Check For and Resolve Discretization Errors” on page 1-20.](#)

### Inside the Example



3D Configuration of the sample SPEED2000 file (EXAMPLE.spd)



Cross sectional view of the SPEED2000 file (EXAMPLE.spd)

The example, in the file EXAMPLE.spd, contains the information for a package consisting of 4 layers. As shown in the figures above, the package contains a signal layer at the top, one power plane, one ground plane, and a signal layer at the bottom. There are two traces, **trace a** on the top signal layer and **trace b** on



the bottom signal layer. The two traces are connected to each other through **via 2**. A current source with a source resistor,  $R_s$ , is connected to **trace a** and **via 1** which is between the top signal layer and the power plane. A termination resistor,  $R_{term}$ , is connected to **trace b** and **via 3** which connects the bottom signal layer and the ground plane.

The current source and the source resistor here represent a 50mA driver. The waveform of the current source is a pulse that has a 100ps low-to-high transition, a 300ps duration, and a 100ps high-to-low transition. We will use SPEED2000 to show you:

- Transient voltages at both the driving and the receiving ends.
- The voltage between the power and ground planes at the center of the package.
- The current through the source resistor  $R_s$ .
- The spatial variation of voltages between the power and the ground planes.


#### NOTE:

In SPEED2000 terminology, anything which is not a circuit element is considered part of the “package”. Package can be an IC package or a printed circuit board.

### ***Running the Example SPEED File – Using the Simulator***

The SPDSIM module takes a .spd format file as input, performs time marching calculations, displays animated waveforms, and performs post data processing.

**STEP 1,** Start SPDSIM and load the example:

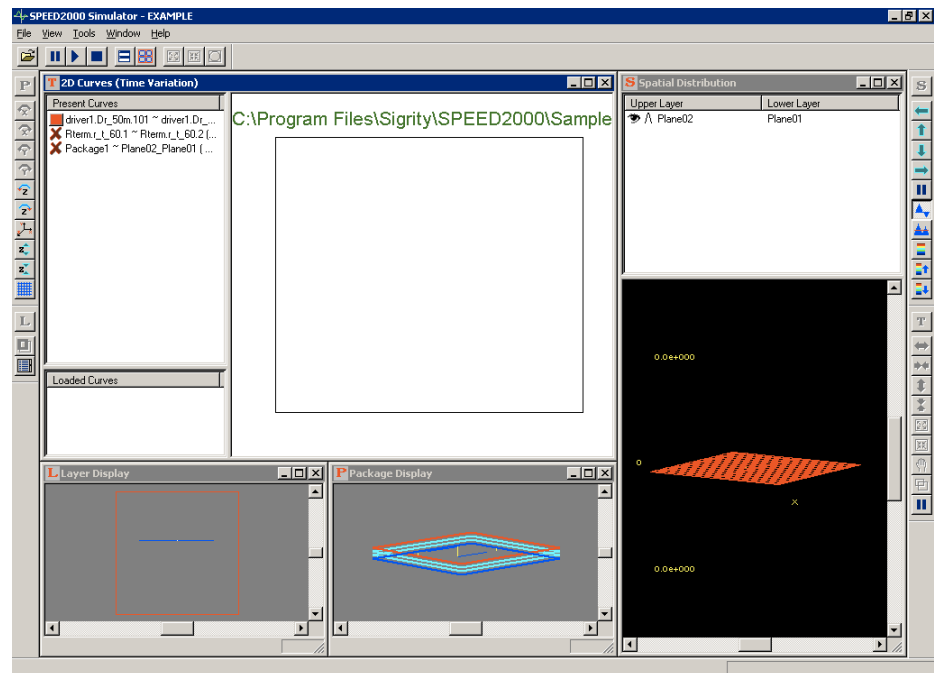
- Launch the SPDSIM module from the Programs menu or double click the SPDSIM icon  in the SPEED2000 program folder to start the simulator.
- Click File: Open from the menu bar. The Open dialog box appears.
- In the Open dialog box, choose the file “EXAMPLE.spd” in the “SPEED2000\Samples” directory, and click the Open button.

#### NOTE:

The default full path name is:  
C:\Program Files\Sigrity\SPEED2000\Samples\EXAMPLE.spd

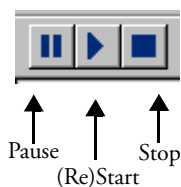
Now your screen should have four windows, as shown in the figure below. (Please see the following note if it does not.)

- The T:2D Curves (Time Variation) window displays 2D curves representing voltages and/or currents versus time. The Present Curves pane is for showing or hiding the curves displayed in the 2D Curves window.
- The P:Package Display window shows a 3D view of the package information within the SPEED2000 file: EXAMPLE.spd.
- The L: Layer Display window shows a 2D view of the sample package information within the SPEED2000 file: EXAMPLE.spd.
- The window titled S:Spatial Distribution displays the animation of spatial variation of voltages between planes/layers. The panes titled upper/lower enable you to select the pairs of layers to be animated.



**NOTE:**

In the event your file does not appear as the figure above, click Window:To Default to display the window configuration above.



**STEP 2,** Start the transient simulation and observe the animated results:

- a. Click the (Re)Start button on the tool bar to start the “time marching” calculations; use the Pause button during the simulation if necessary.

**NOTE:**

The lower-right corner of the screen shows the computation time steps and time elapsed. The speed and length of the simulation can be affected by several variables (such as Mesh, Time Step and Time) which the user can set using the SPDGEN module.

**Tip:**

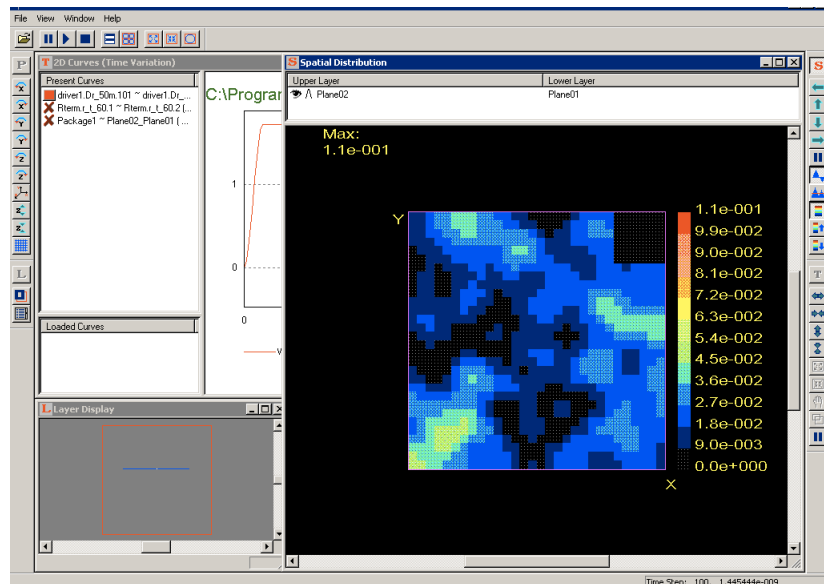
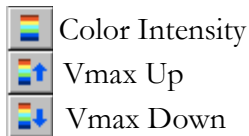
Setting the display mode to 256-color can improve the animation speed in SPD-SIM and reduce the file size for screen capture.

- b. Observe the 2D curves of transient voltages in the 2D Curves window. You can click on the curve name in the curve selection pane to show or hide each curve in the window. The property of the curve can be edited by double clicking on the curve. Right clicking on the curve name also enables you to edit the curve's properties.

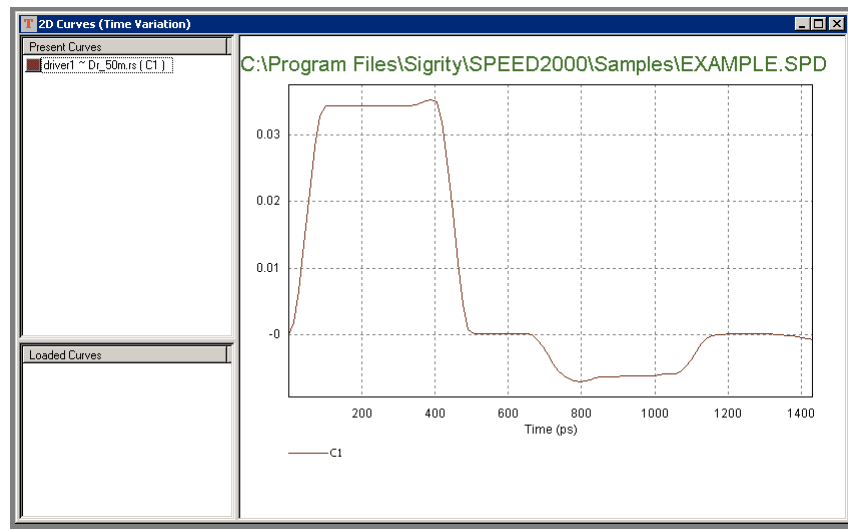
**NOTE:**

Among the three curves in the 2D Curves window, the first two show transient voltages across the driving and the receiving ends, while the third curve shows the voltage fluctuation between the power and ground planes at the center of the package.


- c. Observe the 3D surface plot of the spatial voltage fluctuation between the power and the ground planes in the Spatial Distribution window. Click the Color Intensity button on the right-side tool bar to switch between the wire frame plot and the color-intensity plot. Use the Vmax Up and Vmax Down buttons to adjust the color scale.



- d.** Make sure the 2D Curves window is the current active window, then click Tools: Time Domain: Current from the menu bar. Now, the 2D Curves window displays the variation of current(s) versus time. For the present example, only one curve is displayed, which is the current flowing through the resistor Rs.

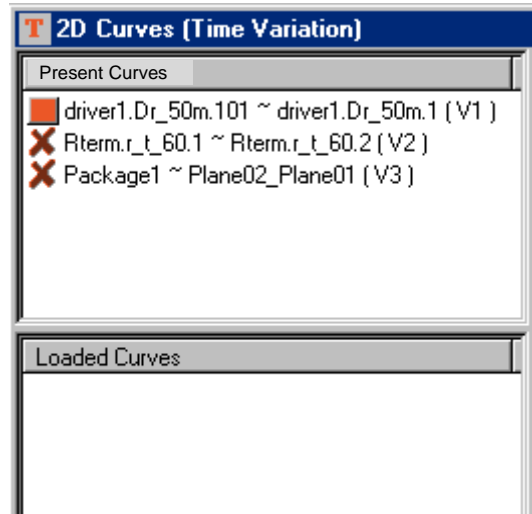


#### NOTE:

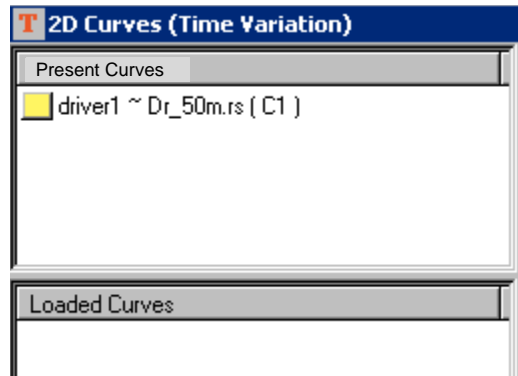
Anytime during the simulation, clicking on the Pause buttons  on the right-side tool bar (one for 3D and one for 2D) will pause the 3D/2D animation, and result in much faster calculation speed.

#### STEP 3, Display and post-process simulation results:

- a.** When the time-marching calculation is finished, make the 2D Curves window as the current active window, then click Tools: Probability from the menu bar. The 2D Curves window now displays the probability distribution of the voltage fluctuations between the power and ground planes.
- b.** Make sure the 2D Curves window is active, click Tools: Time Domain: Voltage to switch back to the transient voltage view in the 2D Curves window. In the Present Curves pane, click on the curve name to make the driving end response (the V1 curve) the only visible curve in the 2D Curves window as shown in the following picture.

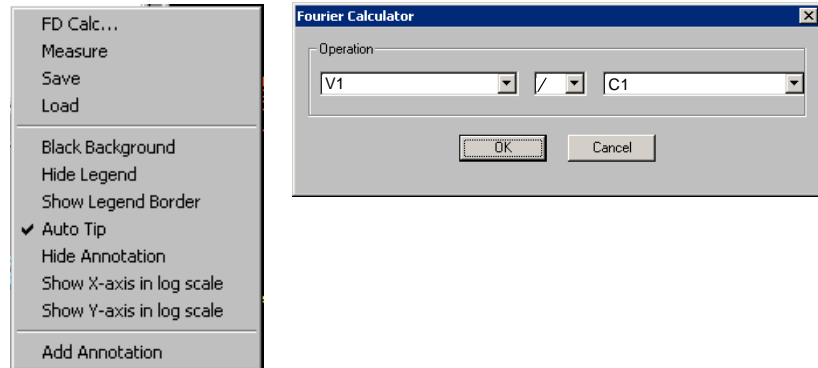


- c. Click Tools: Frequency Domain: Amplitude. The window title changes from 2D Curves (Time Variation) to 2D Curves (Frequency Variation). The curve displayed in the 2D Curves (Frequency Variation) window is the amplitude of the spectrum of the driving end voltage (V1).
- d. Make sure the 2D Curves (Frequency Variation) window is active, click Tools: Time Domain: Current. Now the window title changes back to 2D Curves (Time Variation). The curve displayed in the 2D Curves window and the name of the curve listed in the Present Curves pane are for the current in the source resistor Rs.

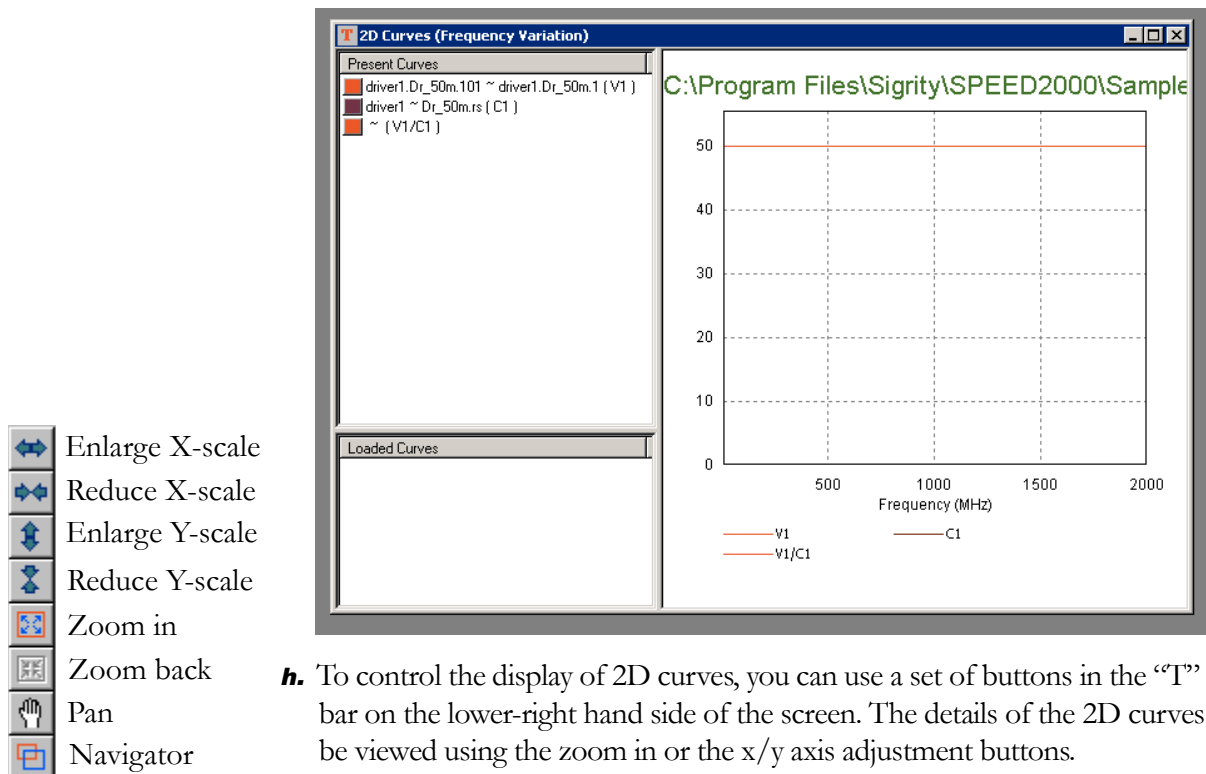


- e. Make sure the 2D Curves window is active, click Tools: Frequency Domain: Amplitude. Now, the 2D Curves (Frequency Variation) window shows two curves. One is the amplitude of the spectrum of the current flowing through the source resistor Rs. The other is the amplitude of the spectrum of the voltage at the driving end.

- f.** In the 2D Curves window, right click and a menu shown below appears. Select F-D Calc..., a Fourier Calculator dialog box appears. Select the operation to be  $V1/C1$ , as shown in the following figure, and click OK.




- g.** The  $V1/C1$  curve now appears in the 2D Curves (Frequency Variation) window, as shown in the following picture. Note that  $V1$  is the voltage across the source resistor,  $C1$  is the current flowing through the source resistor, it can be seen that the  $V1/C1$  curve, which is a horizontal line of a vertical value about 50, agrees very well with the source resistor's value of 50 ohm.



- h.** To control the display of 2D curves, you can use a set of buttons in the “T” toolbar on the lower-right hand side of the screen. The details of the 2D curves can be viewed using the zoom in or the x/y axis adjustment buttons.

- i. 2D curves displayed on windows can be saved by selecting Save in the menu that appears when right clicking in the window. Saved files (filename.cur) can be loaded into SPDSIM at a later time.

**STEP 4,** Re-run the example:

When the simulation is stopped, you can restart it again by clicking the (Re)Start button  on the toolbar.


**STEP 5,** Exit SPDSIM:

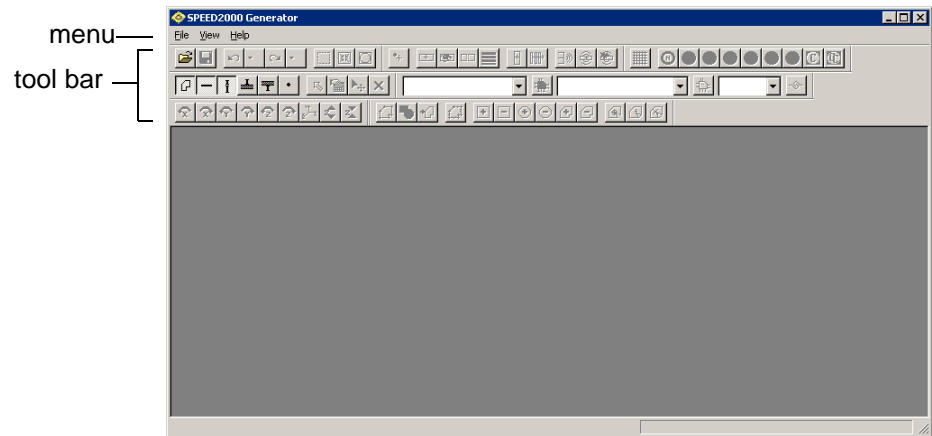
Click File: Exit from the menu bar.

### ***Some Basic Techniques For Using SPDGEN***

In the previous section, you have learned how to use the SPDSIM module to run an example file, EXAMPLE.spd. This section will present some basic technique exercises to familiarize you with procedures used in the SPDGEN module. While SPDSIM loads the file for simulation, SPDGEN is the module that permits you to conveniently edit the package. This makes “what if” analysis a snap.

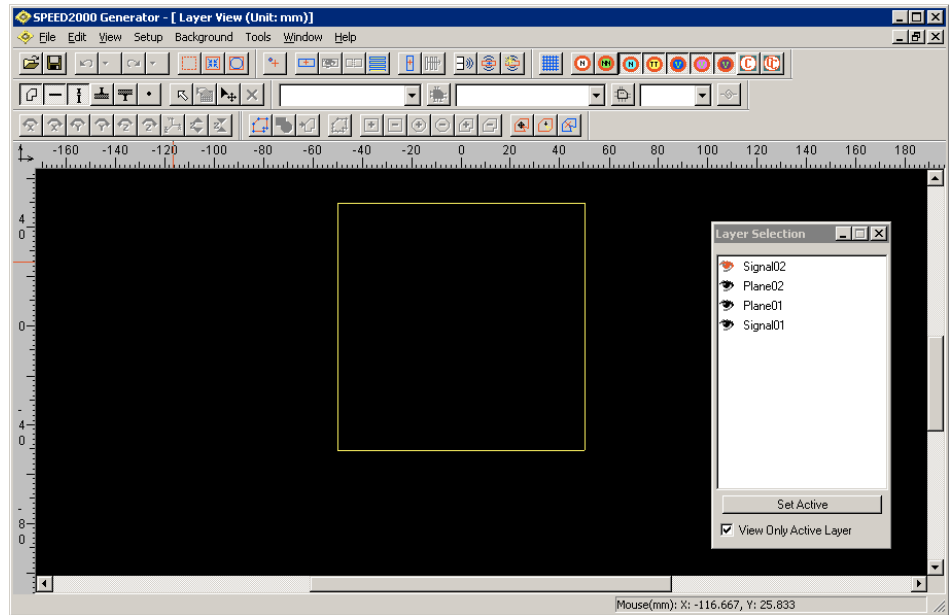
**Exercise 1.** Start SPDGEN

Launch the SPDGEN module from the Programs menu or double click the SPDGEN icon  in the SPEED2000 program folder to start the generator.



**Exercise 2.** Open a New .spd File

Click File: New from the menu bar. Now you should have a screen like the following one. The Layer View window is the place where you can see the “top-down view” of the default package and place components on a selected layer. The layer selection dialog box is shown. The square in the Layer View window outlines the default shape of the package, which is a square 100 mm by 100 mm.





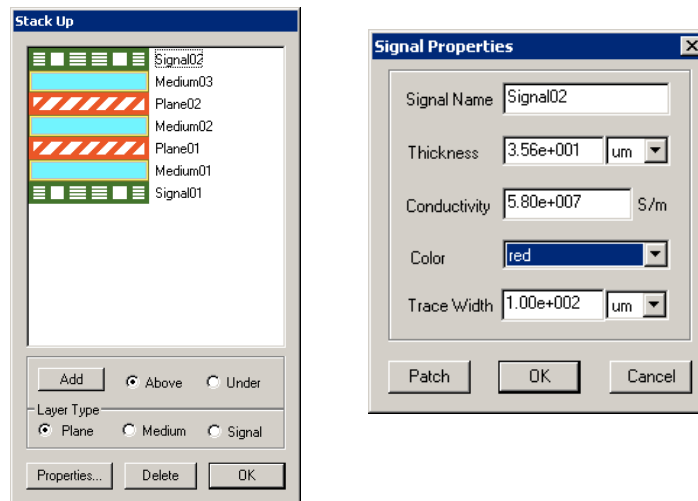
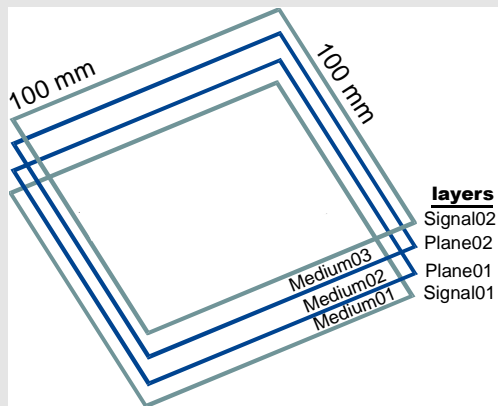
**Exercise 3. Setting Up Layers**

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

**Tip:**

To modify the property of each signal, plane or medium layer, highlight that layer and click Properties or just double-click on the layer.

- a. Click Edit: Stack Up from the menu bar. The Stack Up dialog box appears.
- b. In the Stack Up dialog box, double-click on Signal02, the Signal Properties dialog box appears. Change the Color option for the layer to Red. This will set the default color for all traces on this signal layer to red. Click the OK button to exit the Signal Properties dialog box.
- c. Click the OK button to exit the Stack Up dialog box. Notice the color of the top layer outline, shown in the Layer View window, is now red.
- d. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13](#).

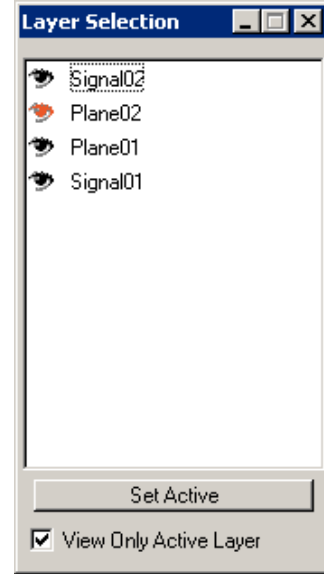
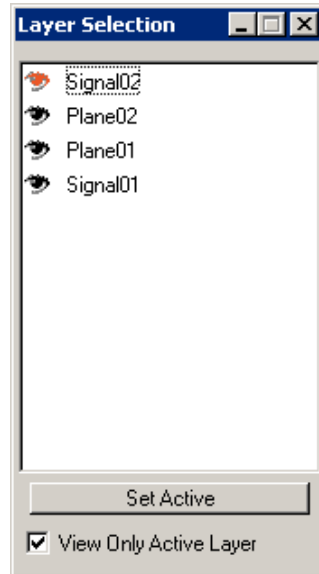
**NOTE:**

**Stack Up versus Layers:** The default setup is a 4 layer structure. The top and the bottom layers, Signal02 and Signal01, are signal layers. There are two solid planes, Plane02 and Plane01 (shown here as power and ground), between the two signal layers. Dielectric media “layers”, in this case Medium01, Medium02 and Medium03, are sandwiched between all package layers. The result (for our example) is a seven layer “stackup” structure. Later you will learn how to change properties of any layer, and add or delete layers

**Exercise 4.** Activate a Layer and Viewing Layers

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

- a. Click View: Layer Selection from the menu bar to bring out the Layer Selection dialog box.



- b. Click the Set Active button in the Layer Selection dialog box, then click Plane02 so that Plane02 is now activated. The eye-icon to the left of Plane02 changes from black to red after layer Plane02 is activated. To perform an operation on a layer, the layer has to be activated first. You can roll up the Layer Selection dialog box by clicking the Minimize button at the upper-right corner.
- c. In the Layer Selection dialog box, uncheck the View Only Active Layer option — this will allow all layers to be viewed on top of each other transparently.
- d. Click on the eye icon to turn on/off the display of each individual layer. When the eye icon is open, this layer is displayed. When the eye icon is closed, this layer is not displayed on the screen.
- e. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13](#).

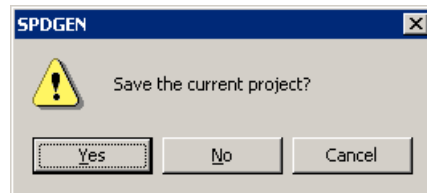
**Exercise 5.** Close the Current .spd File

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

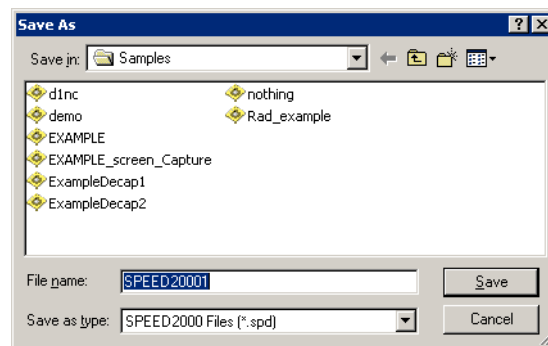
**NOTE:**

In order to interact with the closing and saving User Interface features exercised herein, it is necessary to have performed some operations on either the circuit or package information within the file.

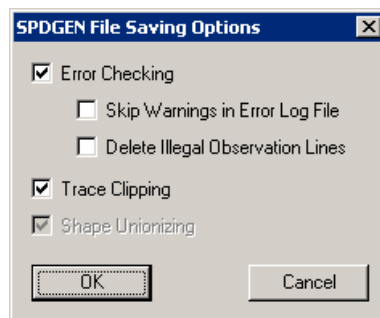
- a. Click File: Close from the menu bar. The Save Warning message box appears since your file has not been saved.



- b. Select Yes to save the file. The Save As message box appears since we have not given a name to the new file. You can choose the name and the location of the file to be stored.



- c. Click Save and the SPDGEN File Saving Options dialog box appears.



- d. Click OK to save the file with the default error checking options and trace/shape processing options selected.

**Exercise 6.** Drawing Traces —also Select/View/Move/Delete Objects, Undo/Redo)

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

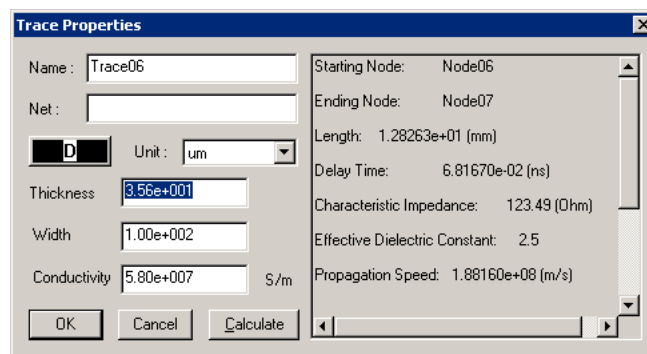


Add Trace

- a. Activate layer Signal01 (as shown in [“Activate a Layer and Viewing Layers” on page 1-12](#)).
- b. Click the Add Trace button on the tool bar. This activates the trace drawing tool.
- c. Move the cursor over the active drawing area of the Layer View window. See the cursor change to the **+** shape cursor.
- d. Click the left mouse button anywhere inside the Layer View window for the starting node of the trace, then move the cursor towards the position for the endpoint (notice a rubber band line follows your cursor) and click once.
- e. A straight line should appear on the screen. You can continue to draw the next segment of the line by moving the cursor to a new location and click again. When you finish drawing the final segment, double click the left mouse button.



- f. Keep on trying until you can easily draw a single or multi-segment trace.
- g. Click the Select button.
- h. Click on the trace that you want to select or drag a rectangular area to enclose multiple traces. The selected traces will be highlighted in cyan color.
- i. Click the Property button to view the trace property.

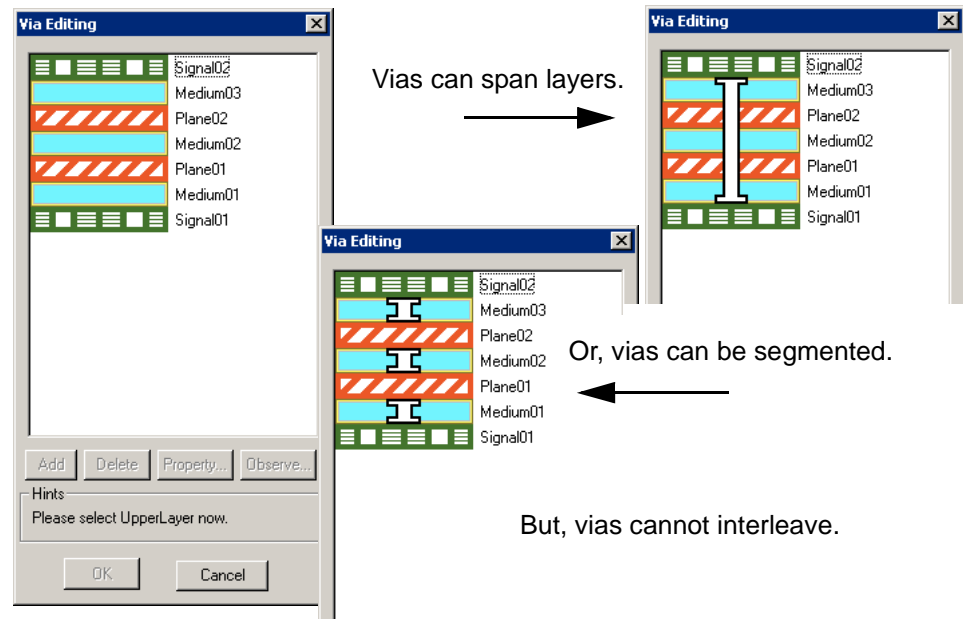


- j. Click OK to exit the Trace Properties dialog box.
- k. To delete the selected trace, simply press the Delete key on the keyboard or click on the Delete button on the menu.
- l. To undo the deletion, click on the Undo button.
- m. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13](#).

**Exercise 7. Adding Vias**

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

- a. Activate the Plane or Signal layer of interest —as shown in [“Activate a Layer and Viewing Layers” on page 1-12](#).
- b. Click the Add Via button on the tool bar. The cursor appears as a **+** as it moves into the Layer View window.
- c. Move the cursor to where you want the via to be located and click the left mouse button once. The Via Editing dialog box appears.



- d. In the Via Editing dialog box, click the left mouse button on the upper package layer and then on the lower package layer that the via connects to.
- e. Vias can span layers or they can be segmented. If more sections are needed, click the Add button in the Via Editing dialog box, and click the upper and lower layers for each via section. (See above)
- f. Repeat the last step until all vertical connections are specified for a specific via.
- g. Click the OK button to exit the Via Editing dialog box.
- h. Practice on adding a few more vias.
- i. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13](#).

**NOTE:**

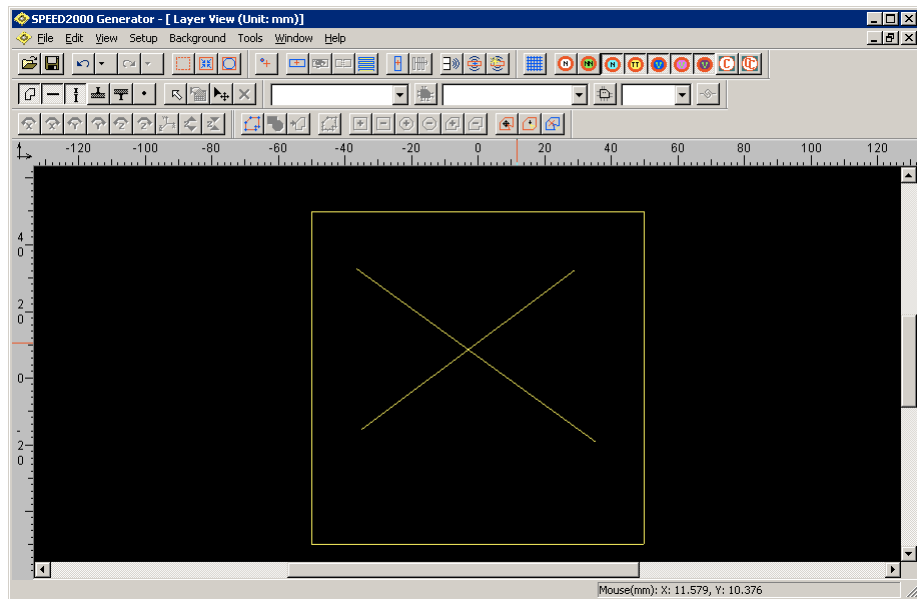
Vias cannot terminate at the medium layer — they can only be connected between signal or plane layers.

**Exercise 8. Check Drawing Errors**

When placing traces on the signal layer, you are indicating electrical connections. That is why SPDGEN, contains error checking procedures to ensure that proper interconnection information is entered. The following procedures demonstrate how to use the error checking functions in SPDGEN to detect a few common drawing errors during the construction of package structures.

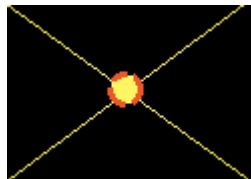
Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

- a. Draw two traces across each other as shown below. (See Exercise [“Drawing Traces —also Select/View/Move/Delete Objects, Undo/Redo” on page 1-14](#) for trace drawing techniques.)



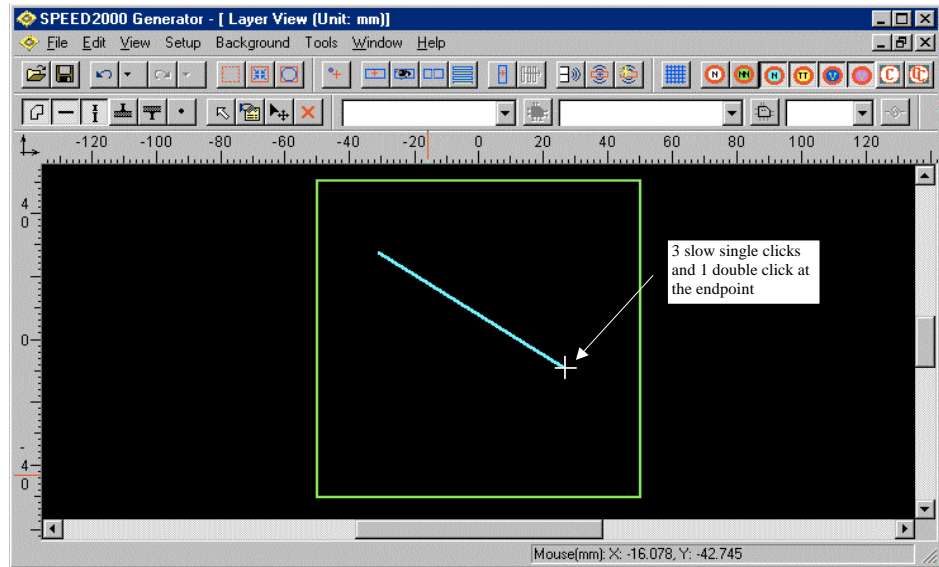
Check

- b. Click the Check button on the tool bar to activate the error checking routines.
- c. An error symbol for Intercepted Traces should appear at the crossing point of the two traces. The cause of the error is that these two traces have no electrical connection though they are graphically across each other. In the next exercise, you will learn how to interconnect traces.

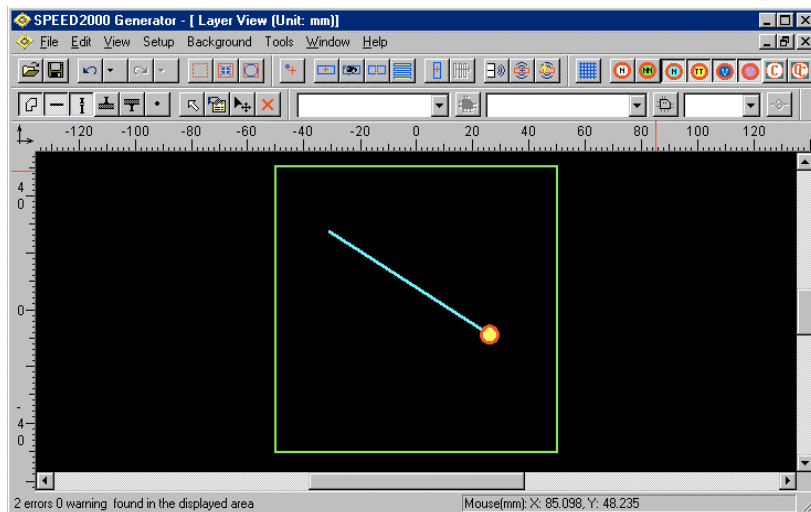


- d. Delete the two traces. (See [“Drawing Traces —also Select/View/Move/Delete Objects, Undo/Redo” on page 1-14](#) )
- e. Click the Add Trace button on the tool bar, move the + shape cursor to the Layer View window.

- f.** Click the left mouse button anywhere inside the Layer View window for the starting node of the trace, then move the cursor towards the position for the endpoint and give 3 slow single clicks and 1 double click at the same location.



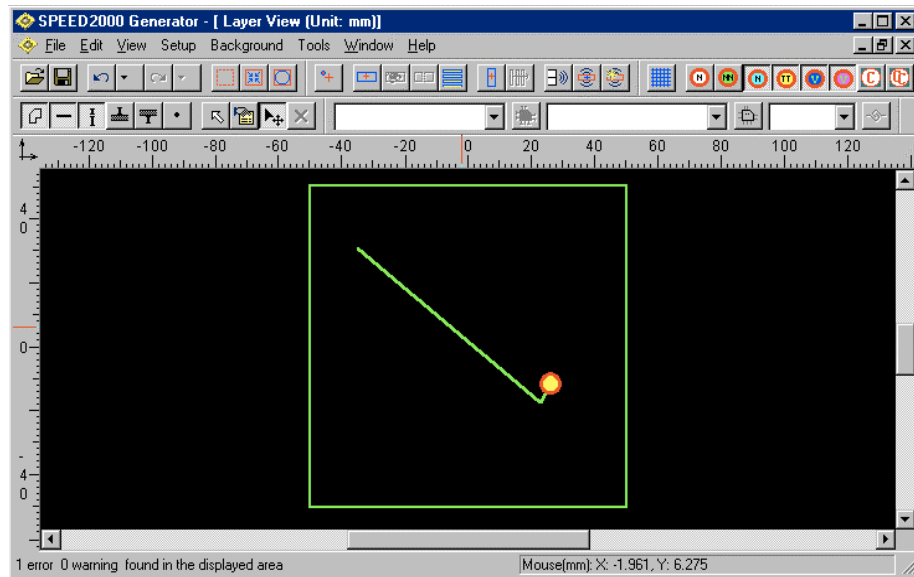
- g.** A straight line similar to the one in the above diagram should appear on the screen. Though this line looks like a normal trace, its ending node conceals several tiny traces overlapping due to the excessive mouse clicks. Very tiny traces generated by excessive mouse clicks are undesirable
- h.** Click the Check button on the tool bar and you will find an error symbol highlights the ending node of the traces.



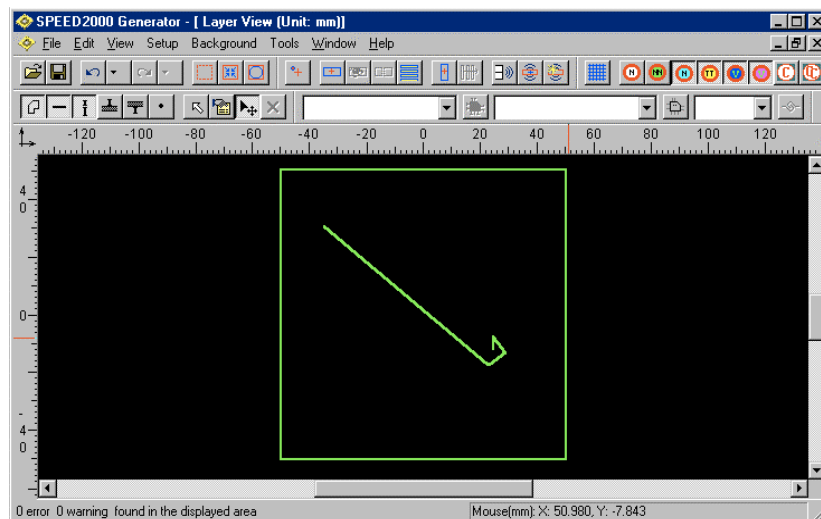
Move

- i.** To further reveal the problem, click the Move button on the tool bar and click on the ending node.

- j. Drag the selected ending node to a new location. Now the excessive trace appears on the screen.



- k. Click the Check button again. If there is still an error symbol, that means there are tiny traces still hidden underneath that node. Select the node highlighted with the error symbol and drag it to a new location.
- l. Repeat the last step until no error indicator will show up when you press the Check error button.



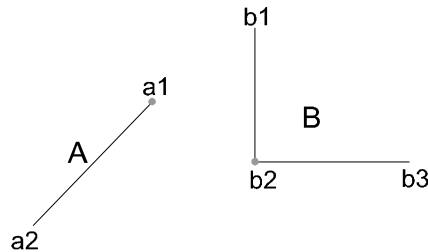
- m. Keep practicing on trace drawing, deleting, node moving and error checking until you feel comfortable with drawing traces (single/double mouse click), moving traces using the Move button and using the Check button.
- n. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13.](#)




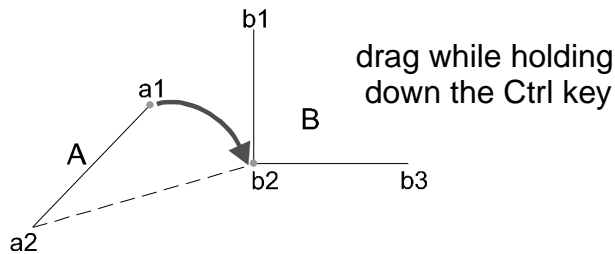
**Exercise 9.** Connect Traces/Merge Nodes

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

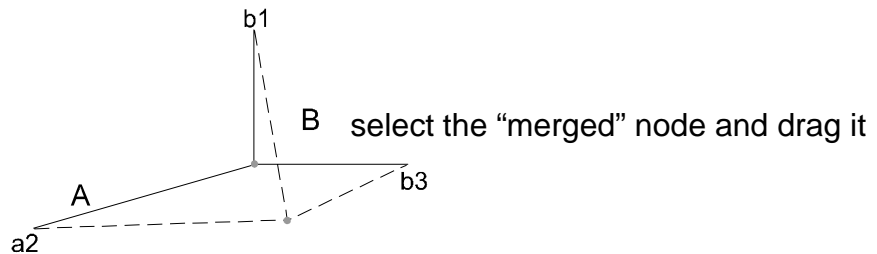
- a. Make sure layer Signal02 is active. (see Exercise [“Activate a Layer and Viewing Layers” on page 1-12](#))
- b. Draw two traces, trace A and trace B, as shown below.



- c. Click the Move  button on the tool bar.
- d. Move the cursor (now activated for a move) over node a1 of trace A.
- e. Press and hold the Ctrl key then —without releasing the Ctrl key—hold down the left mouse button and drag node a1 towards node b2 of trace B. The cursor will display a large crosshair when node a1 closely approaches node b2. When node a1 is close to b2, release the left mouse button.



- f. Now node a1 should be merged with node b2. To test it, select the “merged” node and drag it. Trace A and trace B should move together with the selected node.



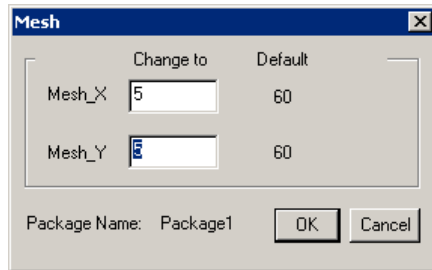
- g. Draw a few traces and use the above “node merge method” to connect them. Later we will use the same technique to connect trace and via.
- h. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13](#).

**Exercise 10.** Check For and Resolve Discretization Errors

For the numerical solution of a differential equation, a continuous problem space is often divided by a numerical mesh. Variables at the discrete mesh points, instead of in the continuous problem space, are then solved by a numerical computation technique. In SPEED2000, a uniform and rectangular mesh is used for the discretization of the continuous voltage and current variables between and on metal planes. For example, during computation, a via is logically snapped to its most adjacent mesh point — and, if the mesh is too coarse, two or more vias might be snapped to the same mesh point. SPEED2000 currently regards such situations as “discretization error”. Use the following procedure to check for and solve “discretization errors”.

Open a new .spd file, if necessary. (see [“Open a New .spd File” on page 1-10](#) )

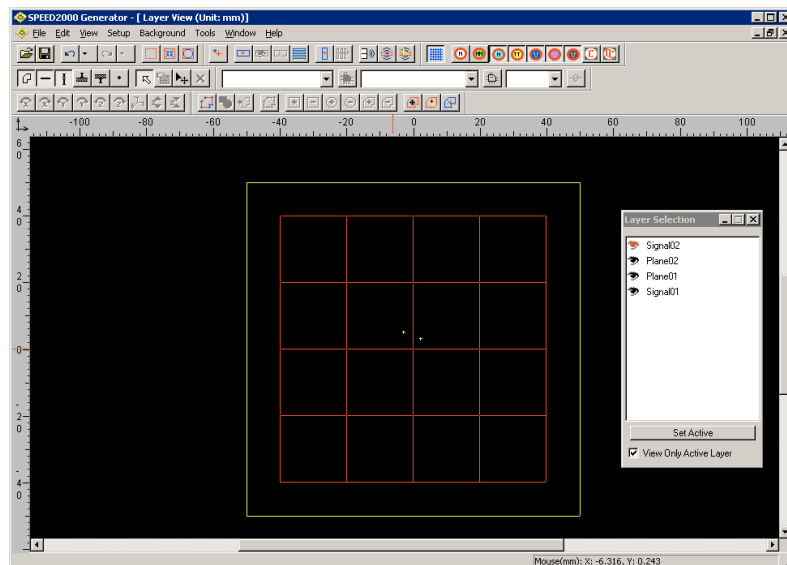
- a. Click Setup:Mesh from the menu bar. The Mesh dialog box appears.



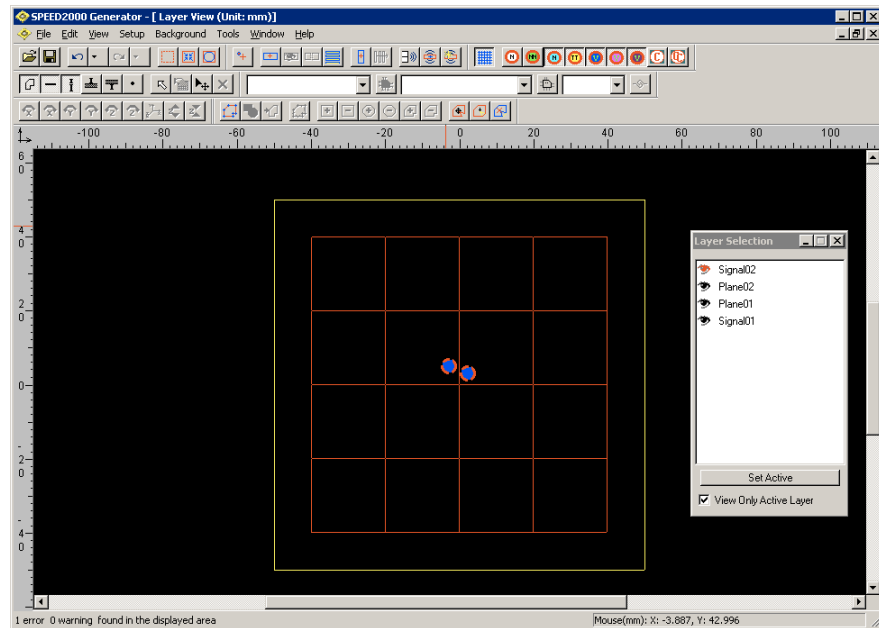
- b. Note that the default value for Mesh\_X and Mesh\_Y is 60, and change the mesh values to 5.
- c. Click the Mesh On/Off button on the tool bar to reveal the mesh.
- d. Draw two vias and position them as shown on the screen. If necessary, see [“Adding Vias” on page 1-15](#)





Mesh On/Off



- e. Click the Check button  on the tool bar. Two error symbols should appear at the via locations, denoting the discretization error.



#### NOTE:

The discretization error is only checked when the mesh is turned on. The Check button, , checks the current active layer, while the Check All Layer Button, , checks all the layers.

- f. To correct the problem, try to set a finer mesh (higher mesh values) until further checking returns no error symbol; Or, you can move the vias — so they are snapped to different mesh points.
- g. This exercise is complete, you may want to close the .spd file —see [“Close the Current .spd File” on page 1-13.](#)

**NOTE:**

Here are some tips on selecting Mesh\_X and Mesh\_Y values:

- The density of the mesh significantly affects the computation speed and memory requirement; however, you will generate a discretization error if two vias are snapped to the same mesh point — therefore, the Mesh\_X and Mesh\_Y values entered must be selected so that two vias cannot be discretized to the same mesh point. Select your mesh values relative to the shortest distance between any two vias in your package. To determine the shortest distance between any two vias in your package use a trial and error method; or, visually examine the package component placement.
- The mesh values must always be integers.
- The minimum value for Mesh\_X and Mesh\_Y is 4.
- The maximum values for Mesh\_X and Mesh\_Y are dependent upon your system resources.
- Sometimes it's helpful to work with the snap feature of SPEED2000 turned on. This helps ensure that object placement will be snapped properly. See [“Setting Up the Display, Drawing and Mesh Environment” on page 4-22](#) for more information.

### ***Building a Simple SPEED2000 File — Using the Generator***

Now that we have mastered [“Some Basic Techniques For Using SPDGEN” on page 1-9](#), let’s return to our Step-by-Step instructions to build a SPEED2000 (.spd) format file using the SPEED2000 SPDGEN module. This section will guide you to use the SPDGEN module to build a file similar to the example file, EXAMPLE.spd. You will construct the package, define the circuits, link the circuits to the package, and specify simulation parameters. The final step will instruct you how to run the file you have created. As before, our goal is to demonstrate How To at this point. Later chapters discuss each of the parameters in detail. Let’s begin:

**STEP 1,** Open a new .spd file:

- a.** Start SPDGEN.
- b.** Click File: New from the menu bar.

**STEP 2,** Edit stack-up:

- a.** Click Edit: Stack Up from the menu bar —the Stack Up dialog box appears.
- b.** In the Stack Up dialog box, double-click on Signal02 —the Signal Properties dialog box appears. Change the Color option for the layer to Red. This will set the default color for all traces on this signal layer to red. Click the OK button to exit the Signal Properties dialog box.
- c.** Click the OK button to exit the Stack Up dialog box. Notice the color of the layer outline, shown in the Layer View window, is now red.

**Tip:**

To modify the property of each signal, plane or medium layer, highlight that layer and click Properties or just double-click on the layer.

**STEP 3,** Exit and save the .spd file:

- a.** Click File: Exit from the menu bar. Select Yes in the SPDGEN dialog box to save your file.
- b.** Name the file, *ex1.spd*. Click Save.
- c.** In the SPDGEN File Saving Options dialog box. Click OK to accept all the default error checking options.

**STEP 4,** Open the saved .spd file.

- a.** Start SPDGEN, if necessary.
- b.** Click File:Open from the menu bar and browse to locate your saved file: *ex1.spd*. Double-click to open the file. Dismiss the error dialog box (Error: There is no circuit linkage) — if it appears.

**STEP 5,** Create the plane shape:

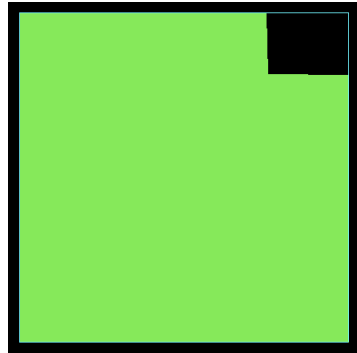
- a.** Click View:Layer Selection.
- b.** Click the Set Active button in the Layer Selection dialog box, then click Plane02 so that Plane02 is now activated. (The Plane02 eye will be red and open.)
- c.** Click Tools: Options: Grid and Unit from the menu bar. The Set Grid, Unit and Snap dialog box appears. This box permits the user to turn on the snap feature

and set the grid units. When the snap feature is “on”, an object (such as, a via, a trace or a shape node) placed will be snapped to either the mesh or grid point that it is closest to. For more information on Mesh, Grid and the “snap to” feature, please refer to [“Understanding The Mesh, The Grid and Snap to:” on page 6-3.](#)

- d.** In the Set Grid, Unit and Snap dialog box, change the display unit to cm and check the Snap To Grid option; then click OK to exit.
- e.** Click View: Show Shape from the menu bar. Now in the Layer View window, the view is changed from an outline to a filled pattern. You should see a square plane which is the default plane shape of 10 cm by 10 cm.
- f.** Click View: Toolbar-Shape from the menu to reveal all the shape-tool buttons.
- g.** Click the Polygon button on the tool bar, then draw a 2 cm by 2 cm rectangle at the upper-right corner of the plane by following the instructions below.



Polygon button

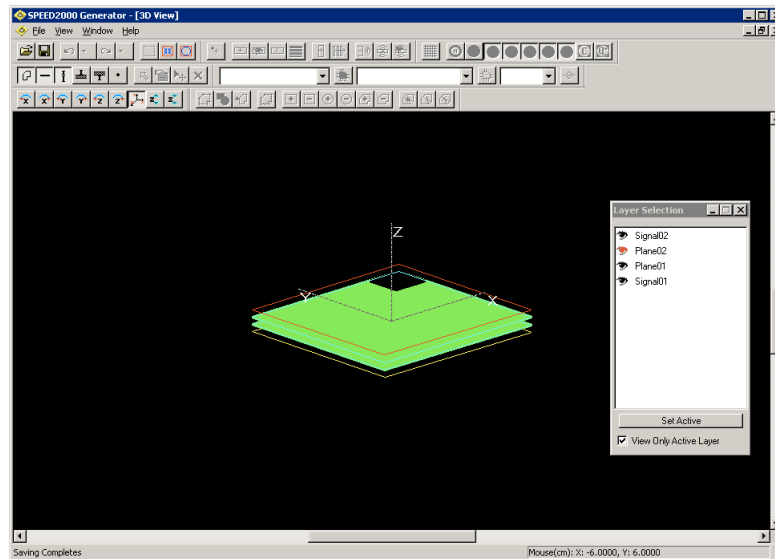


*Click the four vertices of the polygon in the clockwise or counterclockwise order, and **double-click the left mouse button to end the drawing.** This should cut a corner out of the plane as shown left.*

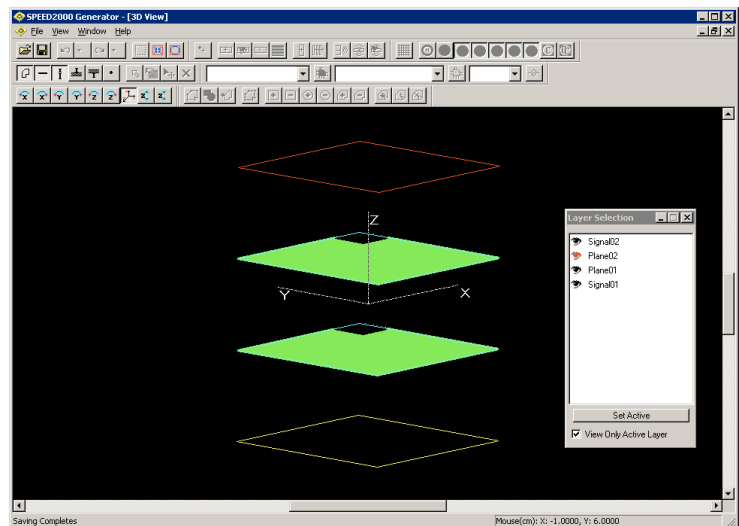
#### NOTE:

If you made a mistake, click the Undo button on the tool bar and try again.

- h.** Click View: 3D View from the menu bar to see the 3D plot of the structure.

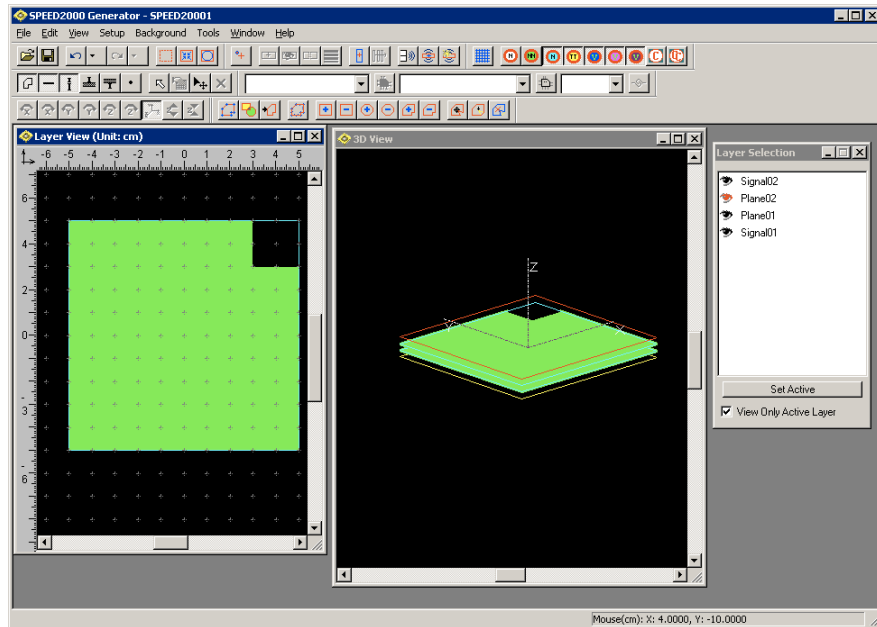


- i.** Click the z-scale buttons to move the planes along the z-axis. Note that shapes on both Plane02 and Plane01 layers are changed, since Plane02 and Plane01 share the same shape definition. More detailed discussions on how to assign shape definitions can be found in [“Working on Shapes” on page 4-29](#).



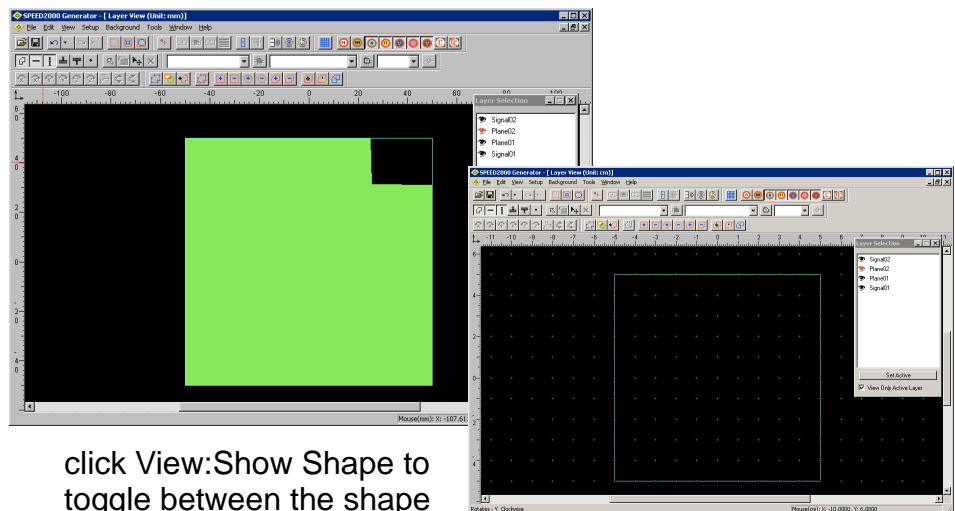
- j.** Click Window:Cascade or Tile to reveal individual window scroll tabs. Click View: Zoom: Fit for each window. Now you should have a screen that looks

like the following one. Note how the menu options change depending upon the window selected.



**STEP 6,** Add vias:

- a. Make sure the Layer View window is the active window, click View: Show Shape from the menu bar to switch from the filled pattern view to the outline view.



click View:Show Shape to toggle between the shape and the outline view



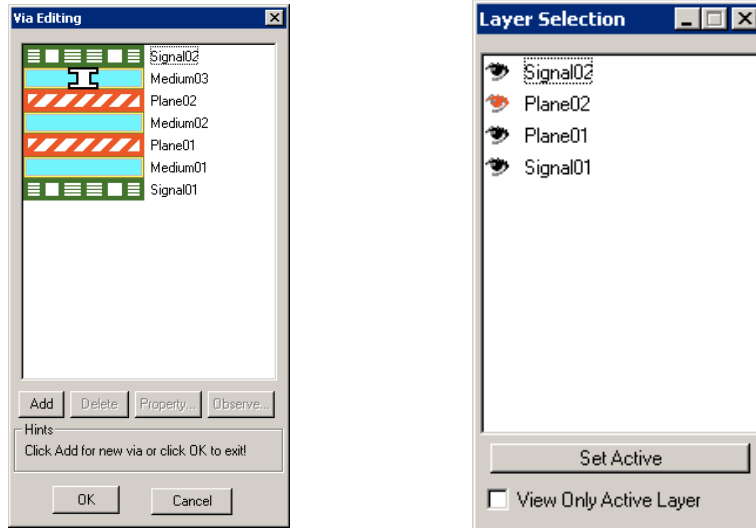
Add Via

- b. Click the Add Via button on the tool bar.
- c. Move the cursor over to the Layer View window. The cursor should transform to a **+** sign. The coordinates of the cursor are shown at the lower-right corner of



the screen. Click at the location with coordinates (-4, 1), a Via Editing dialog box appears.

- d. In the Via Editing dialog box, click Signal02 and Plane02 layers, a via should appear connecting Signal02 and Plane02 Layers.
- e. Click the OK button to exit the Via Editing dialog box. Now via 1 is added.



**Remember to save your work!**

**Click File:Save from the menu bar.**

- f. Move the **+** shape cursor over to the location of coordinates (0,1) and click once. In the Via Editing dialog box, click Signal02 and Signal01. Then click OK. Now via 2, connecting layer Signal02 to layer Signal 01, is added.
- g. Move the **+** shape cursor over to the location of coordinates (4,1) and click once. In the Via Editing dialog box, click Plane01 and Signal01. Then click OK. Via 3 is now added. You may not see via 3 since you are viewing the current active layer Signal02 where via3 does not have a node on it. To see via 3, you can uncheck the View Only Active Layer option in the Layer Selection dialog box.

#### NOTE:


- If you clicked the **+** shape cursor at the wrong location, click the Cancel button in the Via Editing dialog box to exit.
- If you added a via at the wrong location you can undo the addition, or you can delete the via, by clicking the Select button on the tool bar, dragging a box to select the via, then pressing the Delete key on the keyboard to delete the via.

#### STEP 7, Add traces:

- a. Click the Set Active button in the Layer Selection dialog box and click Signal02 to activate the Signal02 layer.
- b. Make sure the Layer View window is the current active window.




Add Trace



- c. Click the Add Trace button on the tool bar, move the **+** shape cursor to the location (-3, 1), click the left mouse button, then move the cursor close to via 2 (which is at the center of the package) while holding down the Ctrl key (when the cursor is close to the via, it should transform to a large hollow cursor with a round tip , this means it is ready to snap onto the existing node), and click the left mouse again. **Double click the left mouse button to complete the trace drawing.** Now trace **a** is done.

**NOTE:**

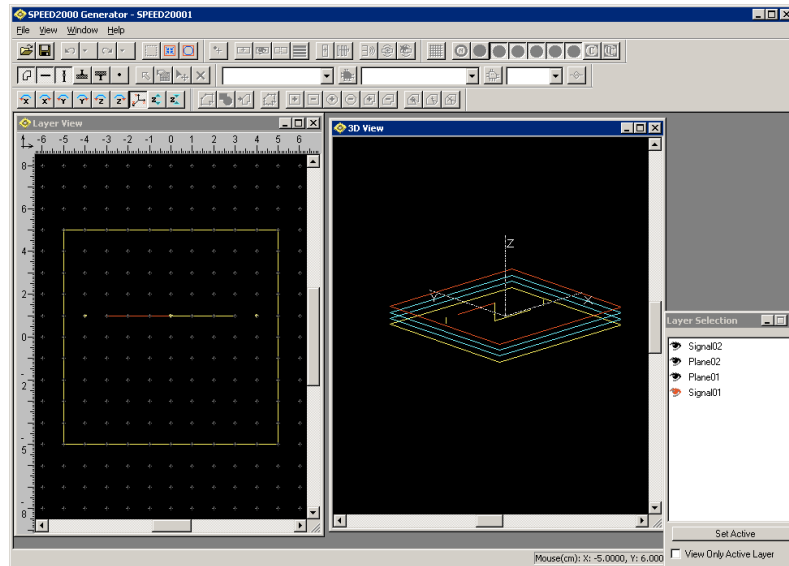
See the figure at the beginning of the chapter, to refresh your memory regarding the Example circuit components (traces a and b and vias 1, 2 and 3). [“SPEED2000 — A Simple Example” on page 1-1](#)

- d. Click the Set Active button in the Layer Selection dialog box and click Signal01 to activate Signal01.
- e. Click the Add Trace button on the tool bar, move the **+** shape cursor to the location (3, 1), click the left mouse button, then move the cursor close to via 2 while holding down the Ctrl key (when the cursor is close to the via, it should transform to a large hollow cursor with a round tip , this means it is ready to snap onto the existing node), click the left mouse again. **Double-click the left mouse button to complete the entering of trace b.**

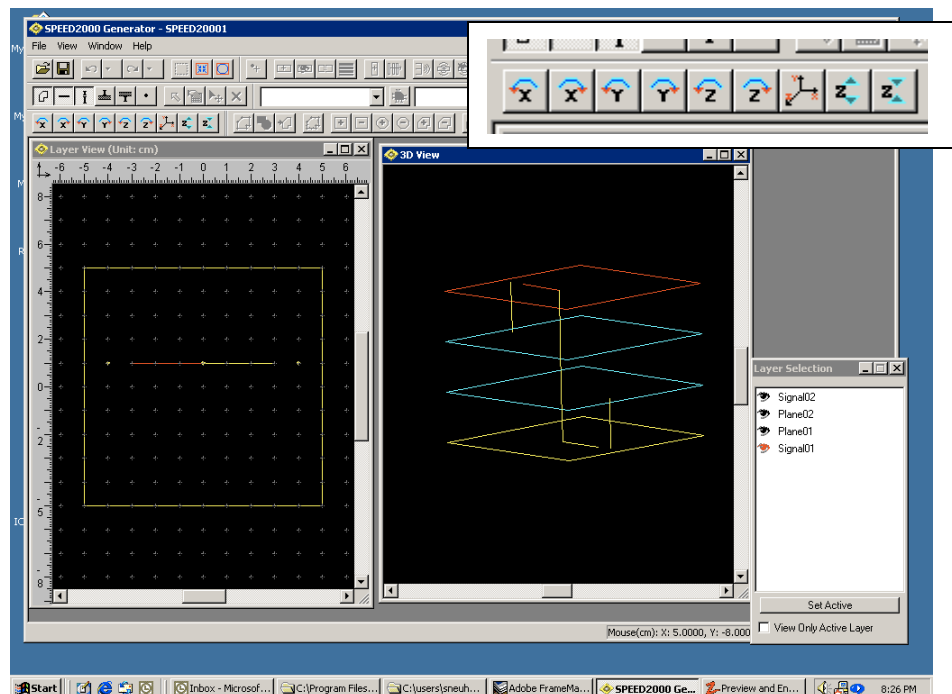
**NOTE:**

If you added a trace at the wrong location, you can undo the addition by clicking the Undo button , or select the trace by using the Select button , then delete it.

- f. Take a look at the 3D view by clicking View: 3D View from the menu bar. The completed package will appear similar as shown below.



- g. Make certain the 3D View window is active, then experiment with the View:Toolbar-3D tool bar, to achieve the view shown below. The Toolbar-3D is discussed in detail in [“Viewing the Structure in 3D” on page 4-4](#)



**Remember to save  
your work!**

**Click File:Save  
from the menu  
bar.**

**STEP 8,** Setup numerical mesh:

- a. Make sure the Layer View window is active, click Setup:Mesh from the menu bar. The Mesh dialog box appears.
- b. In the Mesh dialog box, change the value of Mesh\_X to 30 and the value of Mesh\_Y to 30 also. Click OK to record the new values and exit the Mesh dialog box.

**NOTE:**

The numerical mesh is one of the fundamental computational parameters of the transient simulation — for more information on the mesh settings refer to [“Establishing Computation Parameters” on page 6-2](#). Also see [“Understanding The Mesh, The Grid and Snap to:” on page 6-3](#) to understand the relationship of the grid, the mesh and the “snap to” function. See [“Checking Drawing Errors” on page 4-99](#) to understand how to check for discretization errors.

**STEP 9,** Load circuit definitions:

- a. Make sure the Layer View window is active, click File: Merge Circuit Definitions. The Open dialog box appears.
- b. Enter the circuit file name. In this case, select file DEFAULT.ckt in the C:\Program Files\Sigrity\SPEED2000\Program folder. The DEFAULT.ckt file contains some simple circuit models we are going to use in this example.
- c. Click Open to load the circuit definitions.

**NOTE:**

The Merge Circuit Definitions function opens the file containing the circuit definitions and copies any circuit definitions to the .spd file open; it also copies the circuit definitions to a new file it creates speedfilename.ckt. Circuit definitions for use in SPEED2000 must be specified using the .spd file format syntax. Refer to the .spd File Format Reference for detailed information. This manual is available in the Customer Download area of our website: [www.sigrity.com](http://www.sigrity.com)

**STEP 10,** Specify circuits:

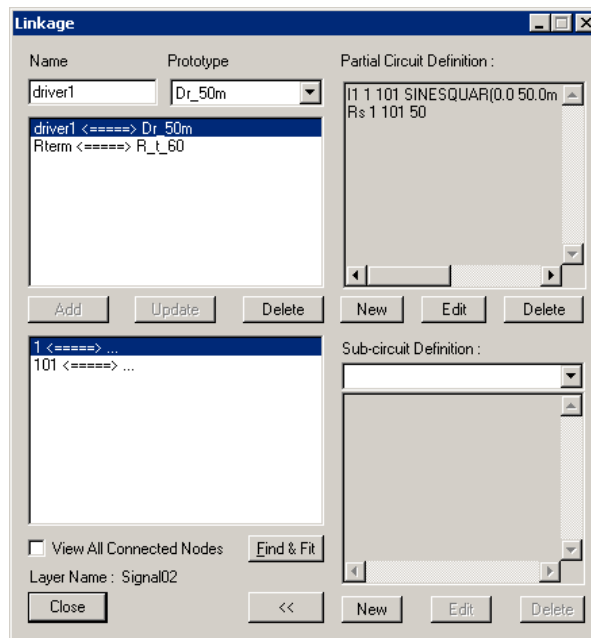
- a. Click Setup: Linkage from the menu bar. A Linkage dialog box will appear. Names of circuit definitions are listed under Prototype. First select the circuit prototype “Dr\_50m” from the list box under Prototype, then type in: *driver1* under the Name of the Linkage dialog box. Click Add. Now we have a circuit “driver1” created using the circuit definition “Dr\_50m”.
- b. Click the >> button at the lower right corner of the Linkage dialog box, the description of the circuit will now be visible in the area of Partial Circuit

Definition. You will find that circuit prototype Dr\_50m is a linear circuit model of a 50 mA driver consisting a 50 ohm source resistor and a 50 mA current source. Detailed information on the SPEED2000 Circuit File Format is available at [“SPD File Format” on page A-1](#). Click the << button to fold in the circuit definition portion of the window.

- c. Select the circuit prototype R\_t\_60 from the list box under Prototype in the Linkage dialog box, then type in: *Rterm* under the Name of the Linkage dialog box, and click Add. Now we have a termination resistor created using the “R\_t\_60” model. R\_t\_60 is a linear resistor of 60 ohm. As before, you can check the definition of circuit prototype R\_t\_60 by clicking the >> button at the lower right corner of the Linkage dialog box. Detailed information on the SPEED2000 Circuit File Format is available at [“SPD File Format” on page A-1](#). Click the << button to fold in the circuit definition portion of the window.

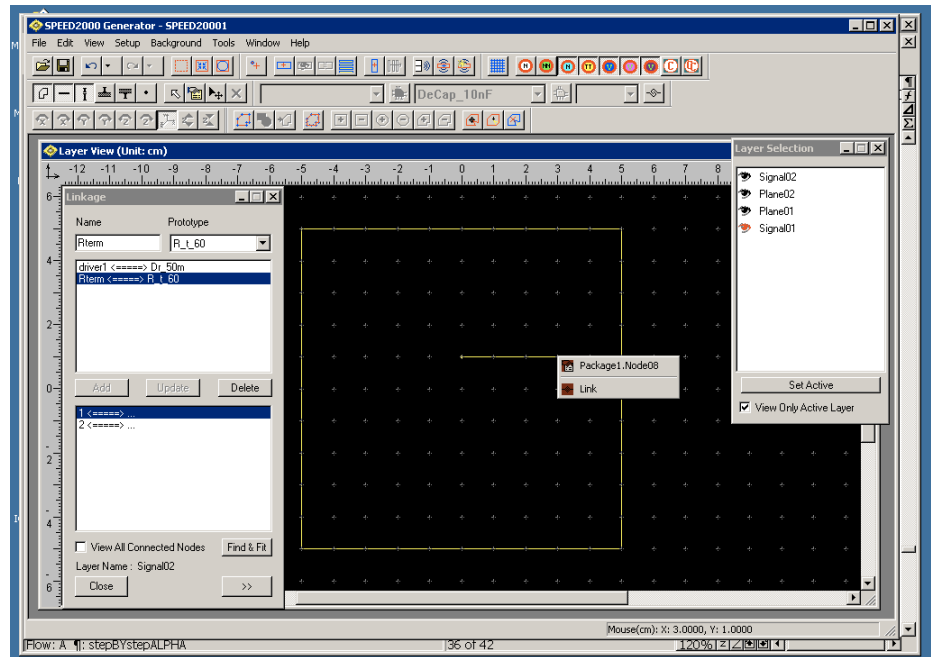
**STEP 11,** Connect circuit to package:

Now we have two circuits specified as shown in the following Linkage dialog box.



- a. Fold in the Circuit definition panel, using the << button, and move the Linkage dialog box to a place where you can see via 3 and trace b in the Layer View window. Make sure Signal01 layer is active.
- b. Highlight the line *Rterm <====> R\_t\_60* in the Linkage dialog box. We are going to connect the 60  $\Omega$  resistor *Rterm* to via 3 and trace b.

- c. Highlight circuit node “1” in the Linkage field of the Linkage dialog box.



- d. Move the cursor over to the right end of trace b in the Layer View window. Click the right mouse button, a small flying menu should appear as shown in the figure above. Click on Link. Now circuit node 1 of “Rterm” is connected to that package node (a green dot appears at the right end of trace b).
- e. Highlight circuit node “2” in the Linkage dialog box. Move the cursor to the Layer View window, and click the right mouse button on via 3. In the flying menu click Link.
- f. Click the Set Active button in the Layer Selection dialog box and click Signal02 to activate Signal02 layer.
- g. Move the Linkage dialog box to a place where you can see via 1 and trace a in the Layer View window. Select the line driver1 <== ==> Dr\_50m in the Linkage dialog box.
- h. Highlight circuit node “1” in the Linkage dialog box. Move the cursor to the Layer View window. Click the right mouse button on via 1, then click Link in the flying menu.
- i. Highlight circuit node “101” in the Linkage dialog box. Move the cursor to the Layer View window, and click the right mouse button on the left end of trace a, then click Link in the flying menu.

- j. Now the termination resistor and the driver are all connected to the package. Click the Close button to exit the Linkage dialog box.

**NOTE:**

If you made a wrong connection between the circuit node and the package node, you can disconnect the linkage as follows. Highlight the linkage pair in the Linkage dialog box, press the Delete key.

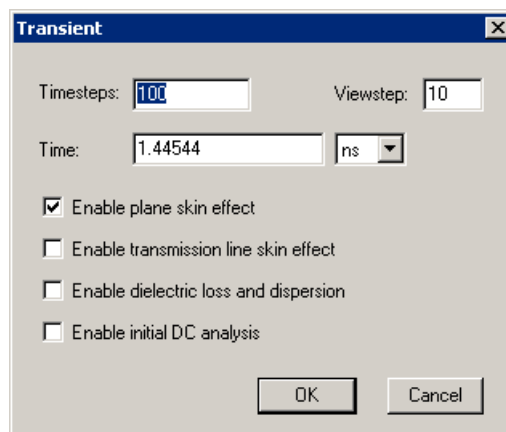
**Remember to save your work!**

**Click File:Save from the menu bar.**

**STEP 12,** Specify simulation parameters:

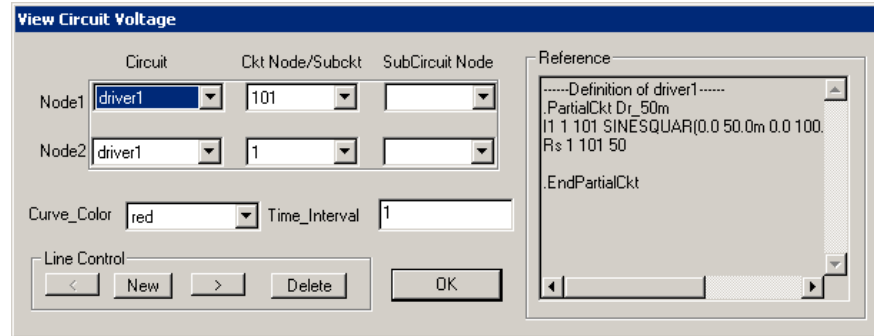
Here we show how to specify several simulation parameters in brief. Detailed information about the simulation parameters can be found [“Establishing Simulation Parameters” on page 6-1](#). Simulation parameters we’ll cover here are:

- Transient—for the specification of computation parameters.
  - View Circuit Voltage—for viewing the dynamic voltage response.
  - View Package Voltage—for viewing the package voltage response.
  - View Current—for viewing the current.
  - View 3D—for viewing the spatial distribution of voltages between planes.
  - Distribution—for viewing the distribution of voltage magnitude between planes.
- a. Click Setup: Transient from the menu bar. The Transient dialog box will appear for the specification of computation parameters. At this time, click the OK button to accept all the defaults. We discuss the simulation parameters in detail within [“Establishing Simulation Parameters” on page 6-1](#).



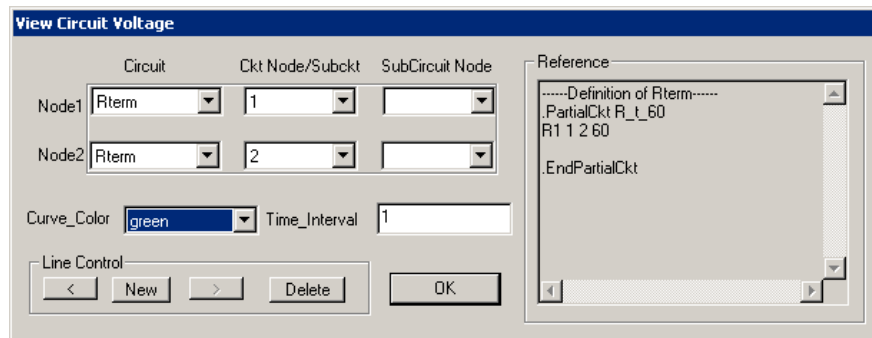
- b. Click Setup: Simulation View: Ckt Voltage from the menu bar. A View Circuit Voltage dialog box appears. Click New to start a new entry.

- c. In the View Circuit Voltage dialog box, two circuit nodes need to be selected for viewing the dynamic voltage response. For Node1, first select *driver1* in the Circuit list box, then select *101* in the Ckt Node/Subckt list box. For Node2, first select *driver1* in the Circuit list box, then select *1* in the Ckt Node/Subckt list box. Select the color for this curve to be red. The completed View Circuit Voltage dialog box looks like the following figure.

**NOTE:**

The definition of the circuit that you selected will appear in the right hand Reference area. Detailed information on the SPEED2000 Circuit File Format is available at [“SPD File Format” on page A-1.](#)

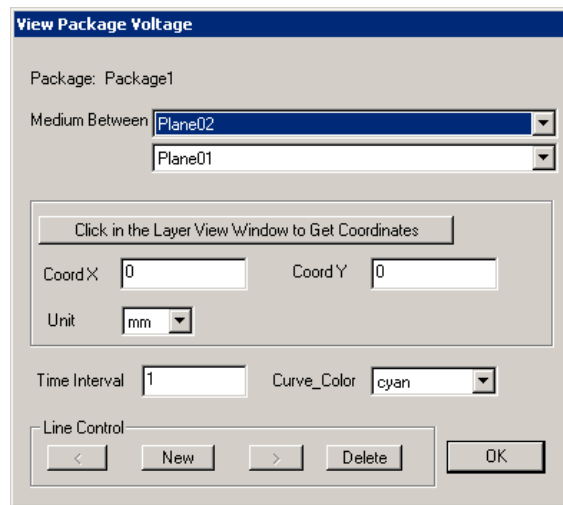
- d. Click the New button in the View Circuit Voltage dialog box to specify another curve. For Node1, first select *Rterm* in the Circuit list box, then select *1* in the Ckt Node/Subckt list box. For Node2, first select *Rterm* in the Circuit list box, then select *2* in the Ckt Node/Subckt list box. Select the color for this curve to be green.



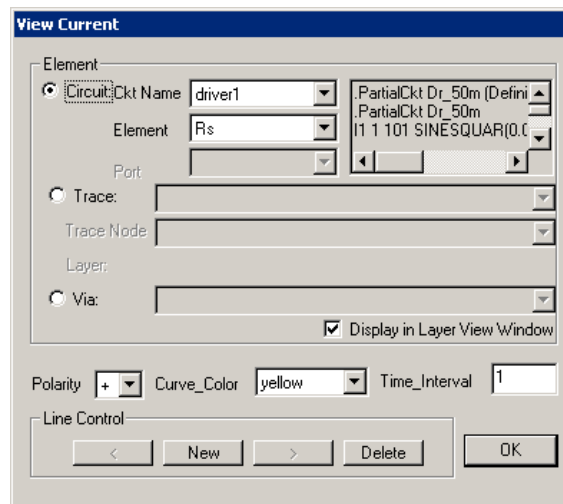
- e. Click OK to exit the View Circuit Voltage dialog box.
- f. Click Setup: Simulation View: Pkg Voltage from the menu bar. In the View Package Voltage dialog box, click New to start a new entry, then select Plane02 and Plane01 in the box next to Medium Between. Select the color for the curve to be



cyan. Your View Package Voltage dialog box should look like the following figure. Click OK to exit the View Package Voltage dialog box.

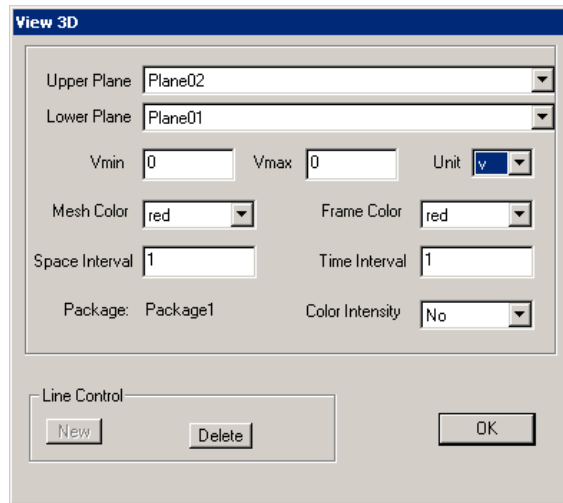


- g. Click Setup: Simulation View: Current from the menu bar. In the View Current dialog box click New to start a new entry. First select *driver1* in the Ckt Name list box, then select *Rs* in the Element list box, and select the color to be yellow. Your View Current dialog box should look like the following figure. Click OK to exit the View Current dialog box.

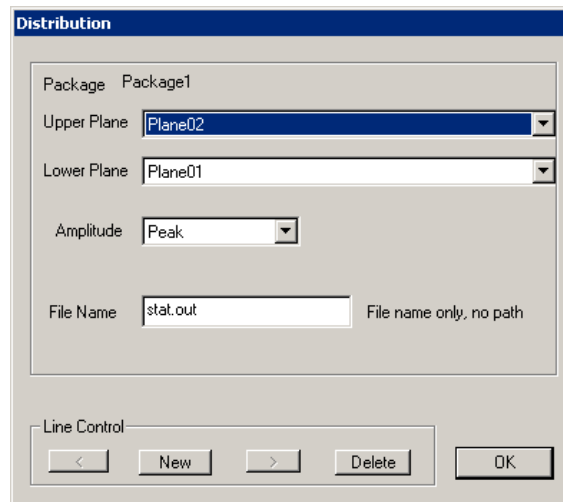


- h. Click Setup: Simulation View: 3D from the menu bar, the View 3D dialog box appears. Click New to start a new entry. Select Plane02 to be the Upper Plane and select the Mesh color and the Frame color to be red. Select the Unit to be

volts. Your View 3D dialog box should look like the following figure. Click OK to exit the View 3D dialog box.



- i. Click Setup: Distribution from the menu. The Distribution dialog box appears. Click New to start a new entry. Select Plane02 to be the Upper Plane and Plane01 to be the Lower Plane. Type in “stat.out” as the File Name and click OK to exit the Distribution dialog box.



**Remember to save your work!**

**Click File:Save from the menu bar.**

### **STEP 13,** Check errors:

- a. Activate layer Signal02, turn on the mesh by clicking the Mesh On/Off button, and then click the Check button.
- b. Correct any errors found. The SPDSIM module will not process a .spd file containing errors indicated during the checking process.
- c. Check errors again — continue checking until all errors are eliminated.
- d. Activate layer Signal01, make sure the mesh is on, and then click the Check button.


- e. Again, correct any errors found.
- f. Check errors again — continue checking until all errors are eliminated.

**STEP 14,** Save the .spd file and Close SPDGEN.

- a. Click Save.
- b. Click File: Exit from the menu bar.

Now you have built a SPEED2000 file (.spd) similar to the EXAMPLE.spd file.

**STEP 15,** Run the file you have just built:

- a. Start the SPDSIM module by double clicking on its icon .
- b. Load “ex1.spd” that you just created with SPDGEN.
- c. Click the (Re)Start button ► to start the simulation. You can follow the instructions described earlier to visualize, process and save simulated results.

**NOTE:**

If you get a warning message, such as “The file contains errors. Please check the error log” during file opening stage, please open ex1\_GENERATOR.err with a text editor (such as Notepad) to view the error. Then load the file into SPDGEN and try to correct the errors.

***Congratulations!***

If you have successfully run the example you just created, you have mastered the basic techniques needed for running SPEED2000. The remainder of the User’s Guide goes into detail on the features of SPEED2000 and how to use them to “Achieve What Others Can’t”.



## SPDGEN Features Overview

SPEED2000 is composed of two executable modules SPDGEN and SPDSIM. This chapter presents an overview of the features, menus and dialog boxes of the SPDGEN module. An overview of the features, menus and dialog boxes of the SPDSIM module is presented in [“SPDSIM Features Overview” on page 3-1](#). This chapter presents an answer to the questions: What are the features of SPDGEN? What is the feature for? Where is it located? and How do I invoke it? A procedural, step-by-step, presentation of *how to use* the features of SPDGEN is available in the following chapters. We provide pointers to specific portions of the procedural descriptions through-out this features overview presentation.

### Menu Option Citation

SPDGEN uses drop-down menus to facilitate User interaction. The menus available change depending upon the Window that is activated. In this chapter, menu options are cited in conjunction with their parent option, the main menu title and the activated window title as a path delimited with colons (:) — SPEED2000 module:window title:menu title:menu option. For example:

### SPDGEN:Layer View>Edit:Stack Up

Where:

“SPDGEN” is the *SPEED2000 module* (SPEED2000 is composed of SPDGEN and SPDSIM executable modules),

“Layer View” is the *window title* (SPDGEN has 2 windows: Layer View and 3D View),

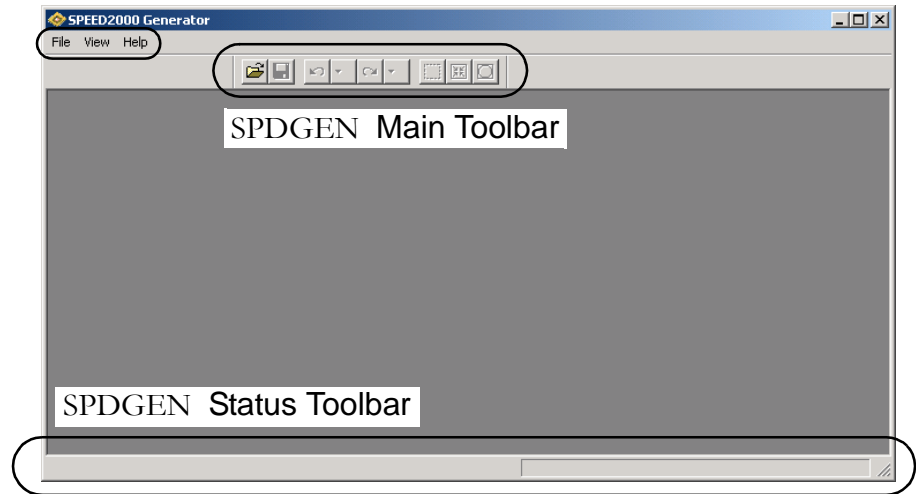
“Edit” is the drop-down *menu title*

and “Stack Up” is the *menu option*.

Initial Main Menu Bar

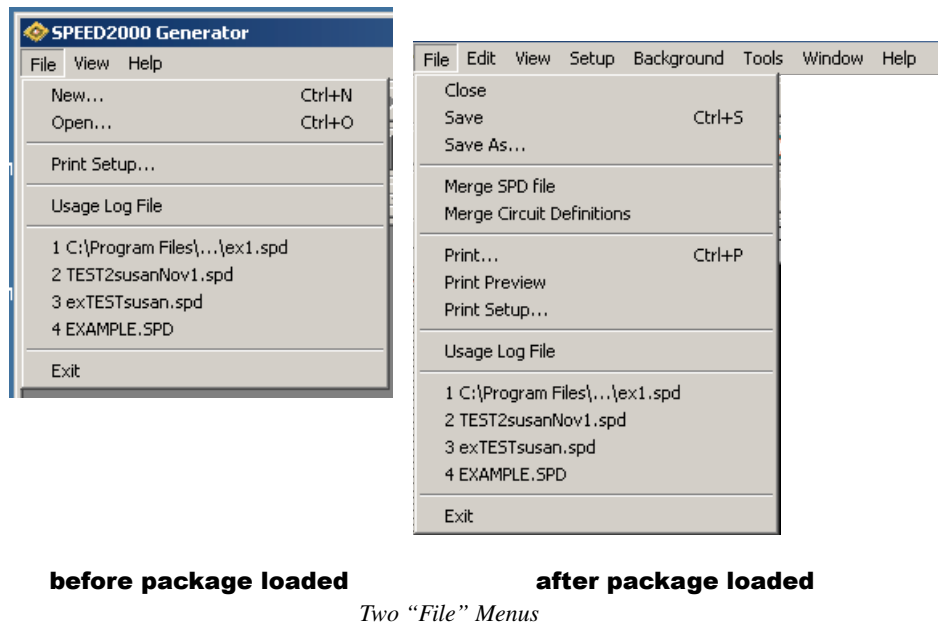
In its simplest state SPDGEN appears as a Graphical User Interface (GUI) workspace with three menu bar options: File, View and Help and the Main (SPDGEN) Toolbar. Other toolbars and the Layer View and 3D View main menus are discussed in the [Layer View](#) and [3D View](#) window sections.

SPDGEN Main Menu



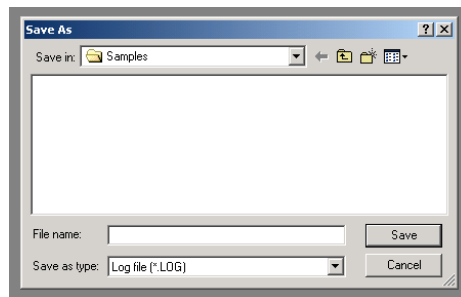
File Menu (before package loaded)

Clicking File from the main menu SPDGEN unfolds the File drop-down menu with options unique to the status "before package loaded". The options that appear on the File drop-down menu are different "after the package is loaded". For more information refer to ["File Menu \(after package loaded\)" on page 2-9](#).



The File drop-down menu contains the following options:

- File:New — Selecting this option invokes the default .spd package to appear in the Layer View window. Please see [The Default SPEED Package](#) for further information on the default package.
- File:Open — Selecting this option invokes the Windows OPEN dialog box, where you can browse (at the system level) to select any SPEED format file (.spd) to open.
- File:Print Setup — Selecting this option invokes the Windows Print Setup dialog box, where you can select the target printer and print job attributes.
- File:Usage Log File — Selecting this option will invoke the Save As dialog box, which permits you to enter the name of a new SPEED session LOG file or select an already existing file. Please see [The SPEED LOG File](#) section for additional information.

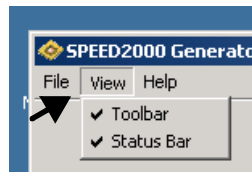


- File:Recent File Listing — Displayed in this section of the drop-down menu are the last four SPEED files accessed — SPDGEN records .ckt, .spd or .log type files accessed.
- File:Exit — If you have performed work on an open file during the previous session, selecting this option will invoke the file saving routines of SPDGEN; if you have not performed work during the session, selecting this option will cause the orderly closeout of the current session and close the SPDGEN module. Please see the SPDGEN [File Saving Options](#) section for further information.

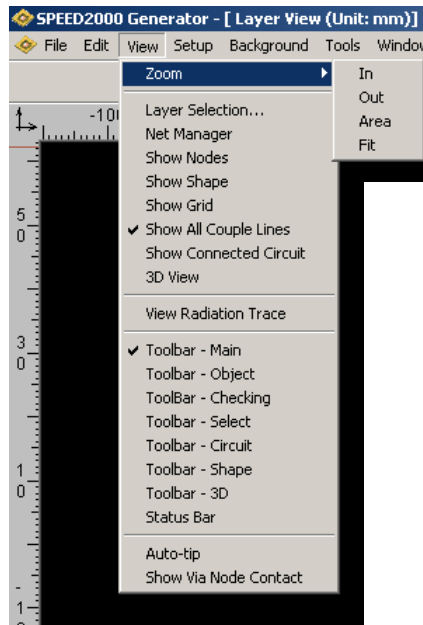
#### **View Menu (before package loaded)**

Clicking View from the main menu of SPDGEN (*before a package is loaded*) unfolds the View drop-down menu. There are three “View” drop down menus. Refer to [“View Menu \(Layer View\)” on page 2-14](#) or [“View Menu \(3D View\)”](#)

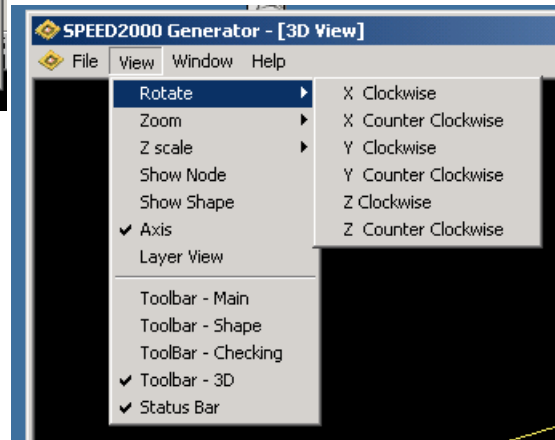
on [page 2-25](#) for further information. The View drop-down menu contains the following options:



**before package loaded**



**after package loaded (Layer View)**



**after package loaded (3D View)**

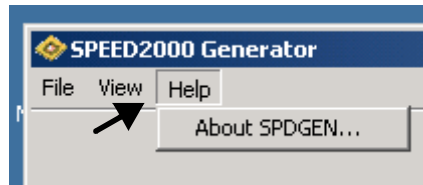
## Three "View" Menus

- View:Toolbar — Toggling this option will display and close the SPDGEN Main toolbar. Please see the [“The Toolbars of SPDGEN” on page 2-26](#) section for further information.
- View:Status Bar — Toggling this option will display and close the SPDGEN status bar at the bottom of the window. This area is used to display system task status messages. For more information on the Status Bar, refer to: [“Status Bar” on page 2-6](#).

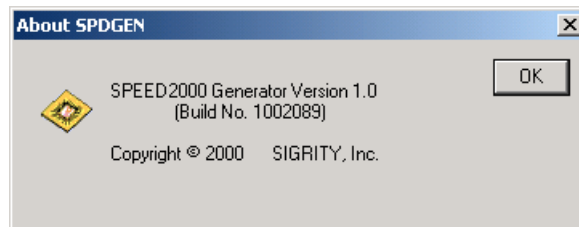


### Help Menu

The Help option drop-down menu contains: About SPDGEN.



- About SPDGEN — Selecting this option will display the About SPDGEN dialog box which details the version and build specific information of the SPDGEN module. If you are anticipating contacting SIGRITY Technical Support, you may find it helpful to display this dialog box and record the version and build information.



### Toolbar - Main (before a package is loaded)

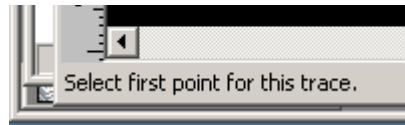
After a package is loaded the SPDGEN:Main toolbar provides convenient access to several functions. In its initial status, before a package is loaded, only the Open function is active. Press the Open button to open an existing SPEED (.spd) format file package for editing. For additional information refer to [“The Toolbars of SPDGEN” on page 2-26](#).



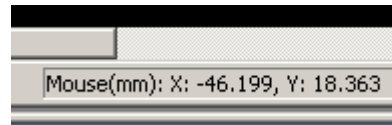
**Status Bar**

The status bar, at the bottom of the application window, is used to display system task status messages, instructions, the purpose of a button or menu option and (when the work space windows contain a package) it records the cursor location for precise positioning of package components.

SPDGEN Status Bar



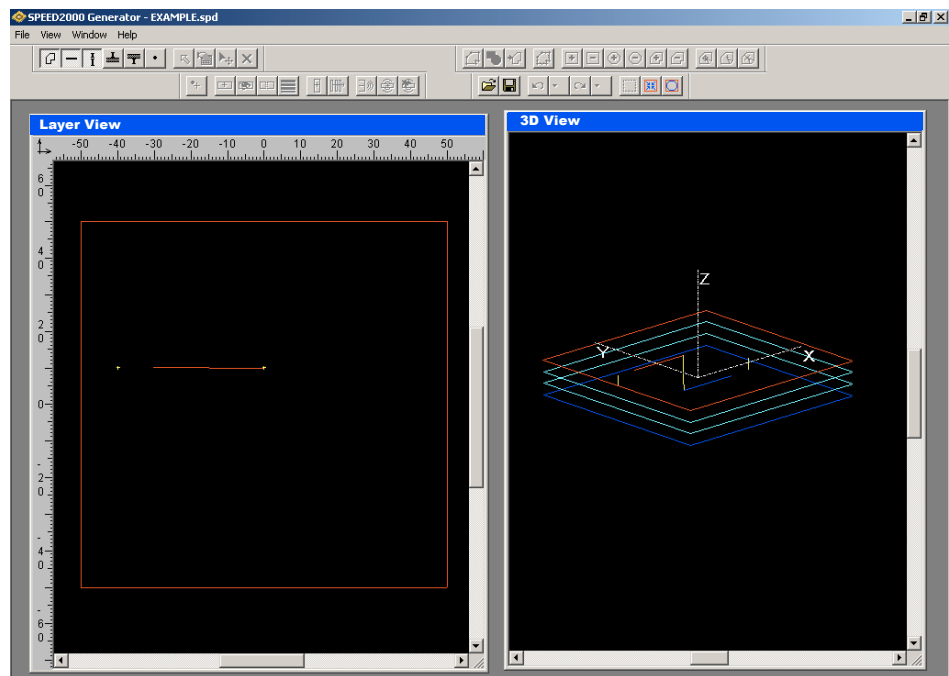
Left side of the Status Bar:  
system task status messages  
and/or the purpose of  
a button or menu option



Right side of the Status Bar:  
records the cursor location for  
precise positioning of package  
components

**SPDGEN Workspace: Layer View or 3D View**

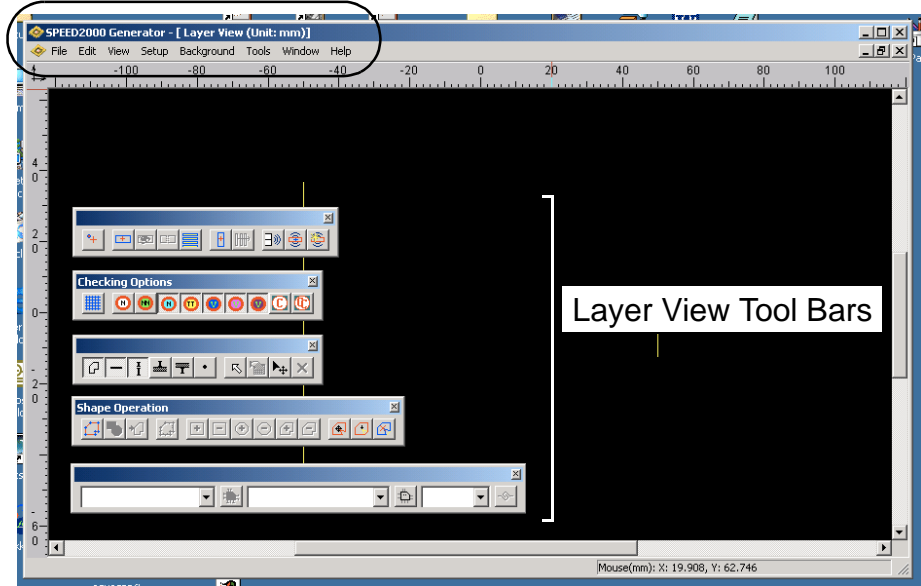
The functions of SPDGEN are available in either the Layer View or 3D View window workspaces. Although conveniently integrated, the menus available and the toolbars are unique for each workspace. Refer to [“The Layer View Window” on page 2-7](#) or [“The 3D View Window” on page 2-24](#). For detailed information on the toolbars of SPDGEN, see [“The Toolbars of SPDGEN” on page 2-26](#).



### ***The Layer View Window***

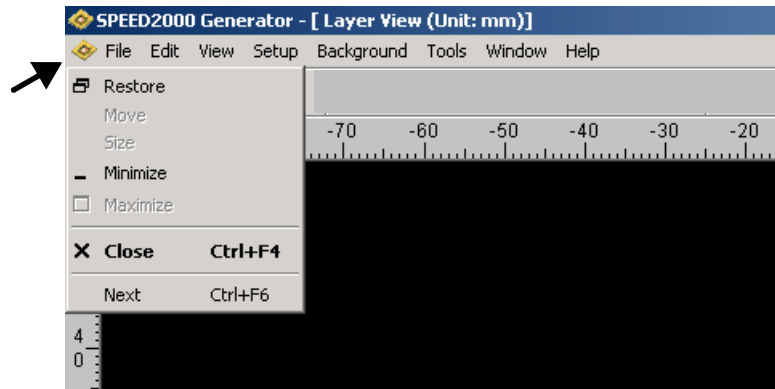
SPDGEN enables the user to graphically work with package components in the Layer View or [3D View](#) Windows. Selecting File:NEW or File:OPEN invokes the Layer View window display of a package. The Layer View Window contains interactive features displayed on a window menu bar and several Layer View window tool bars. A status and information reporting area is available at the bottom of the Layer View window. The Layer View Menu Bar contains several drop down menus: ICON, File, Edit, View Setup Background, Tools, Window and Help. The Layer View Tool Bars provide graphic “tools” which assist the User in the addition, deletion and modification of package elements. This section describes the menu and toolbar options available in the Layer View window.

Layer View Main Menu



**ICON**

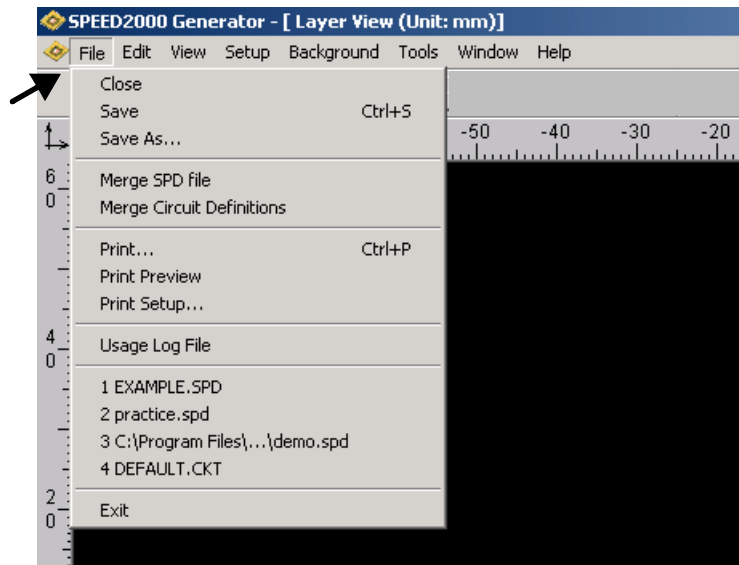
Clicking the SPEED icon in any upper-left window of SPDGEN unfolds the Icon drop-down menu.



- Icon:Restore — Restores the application window to a previous size.
- Icon:Move — Activates the application window for repositioning.
- Icon:Size — Activates the application window for vertical or horizontal resizing on the desktop.
- Icon:Minimize — Minimizes the application to the Quick Launch Toolbar.
- Icon:Maximize — (NA)
- Icon:Close — Closes the current project file, but leaves the SPDGEN application running. If you have performed work in any of the components of the project, you will be prompted to save your work.
- Icon:Next — Moves any displayed and unactive window to the foreground and makes it active.

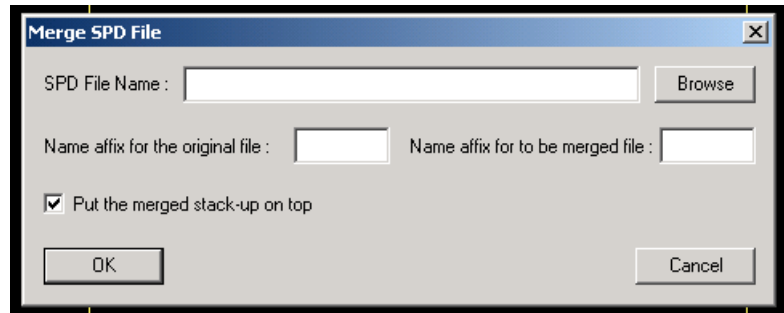
**File Menu (after package loaded)**

Clicking File from the Layer View Menu Bar unfolds the File drop-down menu.

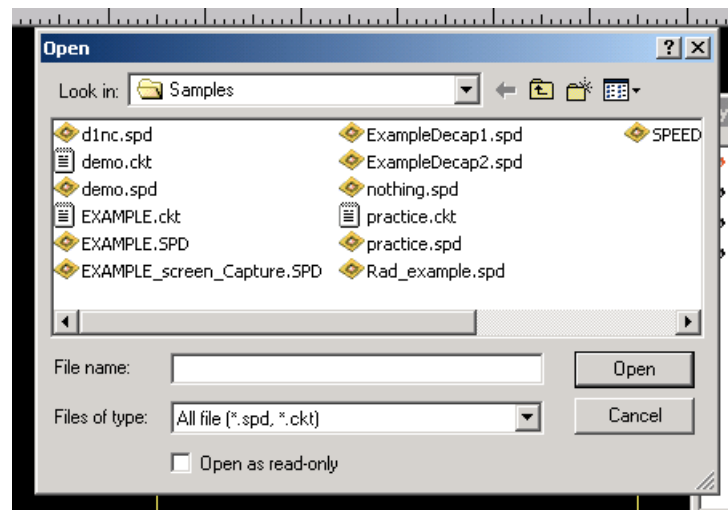


- File:Close — Closes the current project file, but leaves the SPDGEN application running. If you have performed work in any of the components of the project, you will be prompted to save your work. Please refer to the [Exiting and Saving Your Work](#) procedure for detailed information.
- File:Save — Initially, displays the SPEED Save AS dialog box, which permits you to save your work as a .spd format file. After the project file has been saved, at least once, selecting this option displays the [“File Saving Options Dialog Box”](#) on page 4-2 .
- File:Save As — Displays the SPEED Save AS dialog box, which permits you to save your work as a .spd format file with a name different from the current .spd file.

- File:Merge SPD file — You can combine two SPD files together using the File:Merge SPD File function shown below. This feature is useful when you want to connect a package to a printed circuit board to create a more complex 3D model. For more information please refer to: [“Merging Two .spd Files” on page 4-103.](#)



- File:Merge Circuit Definitions — The Merge Circuit Definitions function opens the file containing the circuit definitions and copies any circuit definitions to the .spd file open; it also copies the circuit definitions to a new file it creates speedfilename.ckt. Circuit definitions for use in SPEED2000 must be specified using the .spd file format syntax. Refer to the [.spd File Format Reference](#) document for detailed information on this syntax. This manual is available in the Customer Download area of our web-site: [www.sigrity.com](http://www.sigrity.com)

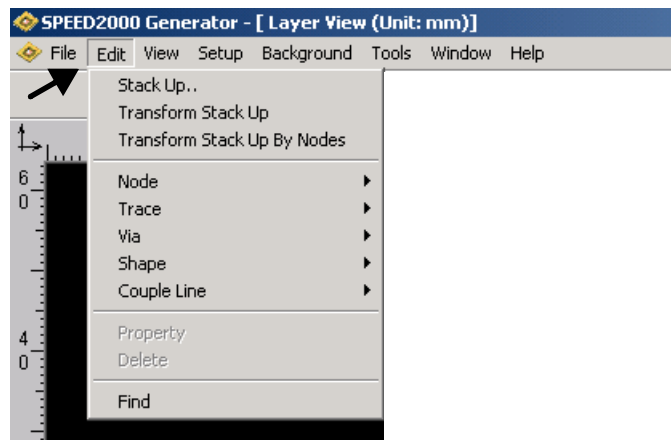


- File:Print — Displays the Windows Print dialog box. From this dialog box you can select the target printer and any print job attributes available for that printer.

- File:Print Preview — Displays the image as it will print to the printer.
- File:Print Setup — Displays a dialog box which permits the user to select and configure any attached network printers.
- File:Usage Log File — Selecting this option will invoke the Save As dialog box, which permits you to enter the name of a usage log file. The log file maintains an administrative records of the usage of the application — it records how many times the application was invoked, by what user on what system. This information may be useful to management to help decide if additional copies of Sigrity products are warranted.
- File:Recent Listings — Listing of the last four .spd files accessed.
- File:Exit — Closes the current project file and EXITS the SPDGEN application. If you have performed work in any of the components of the project, you will be prompted to save your work. Please refer to the [“Closing/Exiting a .spd file:” on page 4-2](#) procedure for detailed information.

### Edit Menu

Clicking Edit from the Layer View menu bar unfolds the Edit drop-down menu.



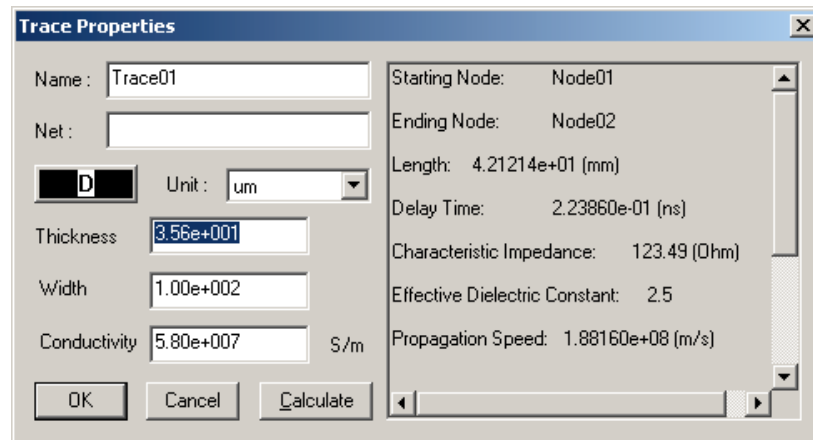
- Edit:Stack Up — The Stack-up dialog box shows the layer information of a package. In this dialog box, you can add or delete three types of layers: Signal layer — on which metal signal traces or patches are placed; Plane layer — solid metal planes of various shapes; Dielectric layer — insulating material separating metal layers. For more information, refer to [“Setting Up the \(Layer\) Stack-up” on page 4-7](#).
- Edit:Transform Stack Up — This dialog box enables the User to move and rotate a portion of a package. It can be useful after merging two packages.

For more information, refer to [“Moving and Rotating a Package” on page 4-105](#).

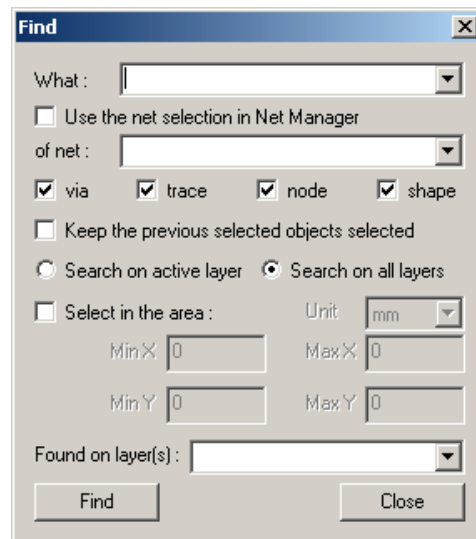
- Edit:Transform Stack up By Nodes — This dialog box enables the User to move and rotate a portion of a package. It can be useful after merging two packages *if the TO and FROM nodes are known*. For more information, refer to [“Moving and Rotating a Package” on page 4-105](#).
- Edit:Node — The Edit:Node option further differentiates to Add and Link Selected Nodes. Nodes define specific locations on the package. For example, the end locations of traces and vias are defined as package nodes. Detailed instruction on how to use node tools can be found [“Using Node Tools” on page 4-25](#).
- Edit:Trace — The Edit:Trace option further differentiates to Add, Split, Observe, Show Selected Traces’ Nodes and Radiation Select. The trace tools can only be applied to signal layers. Therefore, always activate a signal layer before working on any traces on that layer. For detailed information refer to [“Working on Traces” on page 4-91](#).
- Edit:Via — The Edit:Via option further differentiates to Add, Copy To and Via Radius Map. A via provides interconnection between layers in the vertical direction. For more information refer to [“Working on Vias” on page 4-68](#).
- Edit:Shape — Metal planes/patches can have various shapes. The shape of a metal plane is described by a shape definition. Several planes/patches can share one shape definition. To change the shape of a plane/patch, one actually changes the shape definition assigned to the plane/patch layer. The shape tools can be applied to plane layers or patches on signal layers. For more information refer to [“Working on Shapes” on page 4-29](#).
- Edit:Couple line — When traces are identified as coupled lines, the crosstalk between these lines will be calculated during the transient simulation. Coupled lines will be treated as multi-conductor transmission line, whereas an isolated trace is modeled by the single transmission line algorithm. For more information refer to [“Working on Coupled Lines” on page 4-95](#).
- Edit:Property — Select any object and click Edit:Property. The object’s current properties will be reported (in the Properties dialog box) and many



attributes are editable. For more information refer to [“Modifying Object Properties” on page 4-103.](#)

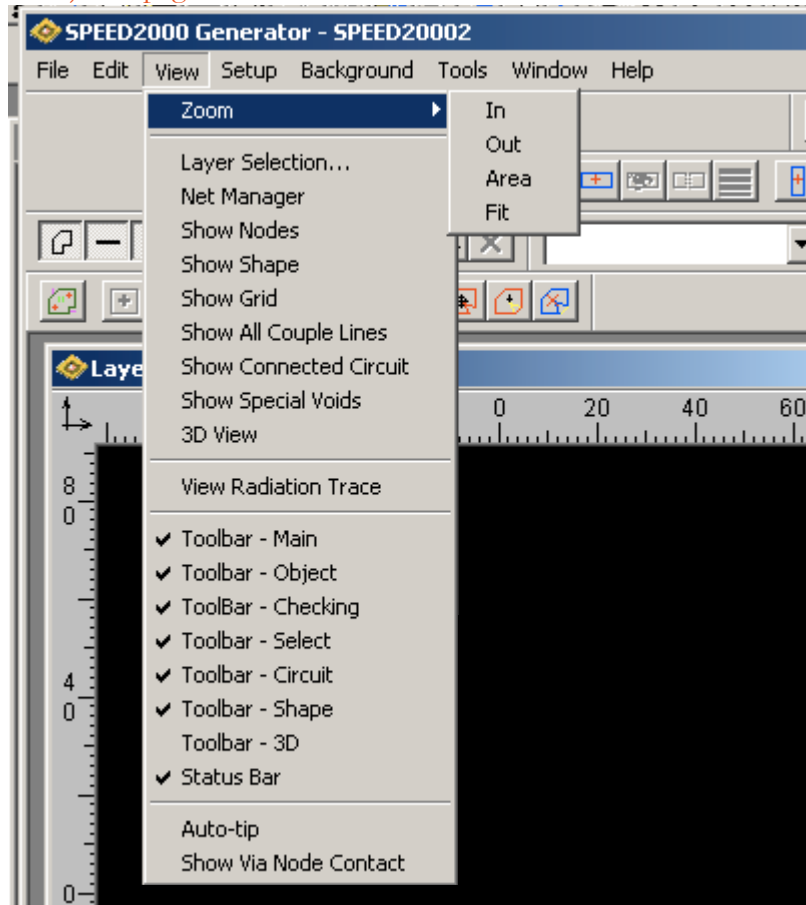


- Edit:Delete — Select any object and click Edit:Delete. The selected object will be deleted.
- Edit:Find — Using the Find dialog box you will be able to search for objects in the .spd project file. Our sophisticated Find function permits you to enter several kinds of search criteria to make searches productive and efficient. For more information refer to [“Finding Objects” on page 4-113.](#)



**View Menu (Layer View)**

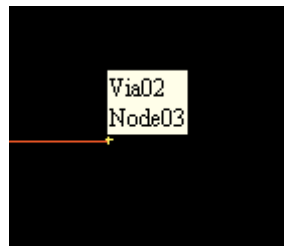
Clicking View (after a package is loaded) from the Layer View menu bar unfolds the View drop-down menu. Compare the [“View Menu \(before package loaded\)” on page 2-3](#) for additional information.



- View:Layer Selection(Layer View) — Use the Layer Selection dialog box to control which layers can be seen and which layer you want to work on in the Layer View window. For more information, refer to [“Viewing Layers and the Layer Selection Dialog Box” on page 4-5](#).
- View:Net Manager(Layer View) — For more information, refer to [“Net Management” on page 4-15](#).
- View:Show Nodes(Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).
- View:Show Shape(Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).
- View:Show Grid(Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).

- View:Show All Couple Lines(Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).
- View:Show Connected Circuit (Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).
- View:Show Special Voids (Layer View) — For more information, refer to [“Showing Objects” on page 4-20](#).
- View:3D View (Layer View) — Select this option to invoke the 3D View window and display the 3D view of the package. For more information refer to [“The 3D View Window” on page 2-24](#) or [“Viewing the Structure in 3D” on page 4-4](#).
- View:View Radiation Trace (Layer View) — Use this option to identify traces to be considered for radiation calculations. For more information refer to [“Selecting Traces and Edges for Radiation Calculation” on page 8-1](#).
- View:Toolbar-Main (Layer View) — Select this option show/hide the Main toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-Object (Layer View) — Select this option show/hide the Object toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-Checking (Layer View) — Select this option show/hide the Main toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-Select (Layer View) — Select this option show/hide the Checking toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-Circuit (Layer View) — Select this option show/hide the Circuit toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-Shape (Layer View) — Select this option show/hide the Shape toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Toolbar-3D (Layer View) — Select this option show/hide the 3D toolbar. For more information refer to [“The Toolbars of SPDGEN” on page 2-26](#).
- View:Status Bar (Layer View) — Select this option show/hide the Status bar. For more information refer to [“Status Bar” on page 2-6](#).

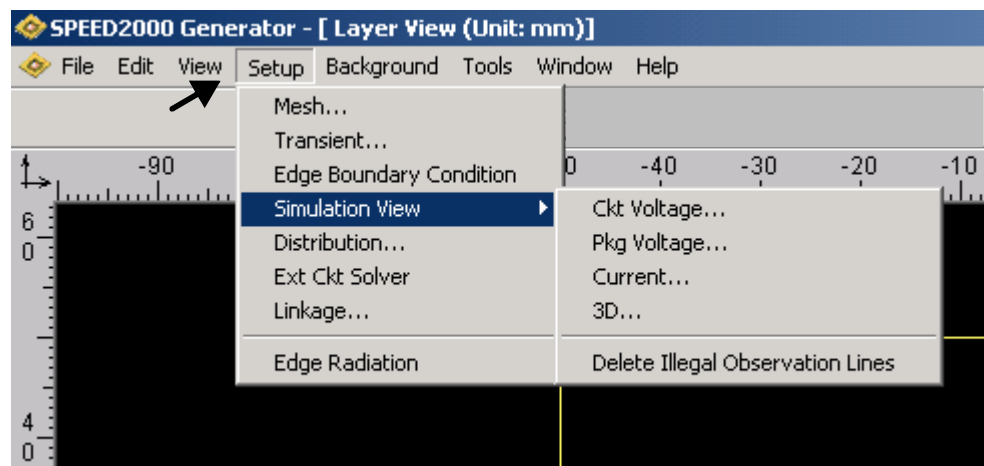
- View:Auto-tip (Layer View) — Use this option to toggle a pop-up informational label for the objects of the package on/off.



- View:Show Via Node Contact (Layer View) — For more information, refer to [“Showing Objects” on page 4-20.](#)

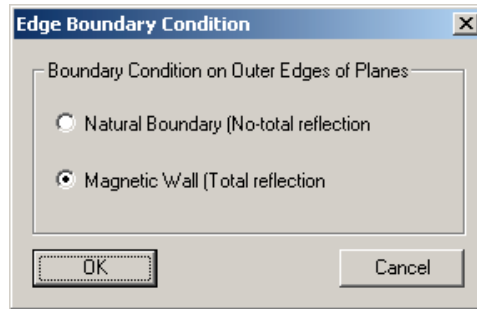
## Setup Menu

Clicking Setup from the Layer View menu bar unfolds the Setup drop-down menu.



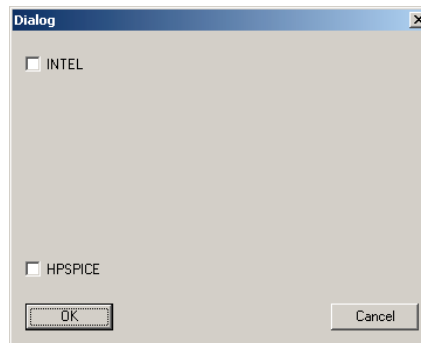
- Setup:Mesh — Select this option to access the Mesh dialog box. A uniform and rectangular mesh is used for the numerical discretization of voltages and currents between and on planes. For detailed instructions on setting the mesh, refer to [“Establishing Computation Parameters” on page 6-2.](#)
- Setup:Transient — The user can adjust several transient simulation parameters using the Transient dialog box. For detailed instructions on setting the Transient simulation parameters, refer to [“Establishing Transient Simulation Parameters” on page 6-5.](#)
- Setup:Edge Boundary Condition — The Edge Boundary Condition enables the User to simulate radiation loss at edges of metal

planes. For information, refer to [“Edge Boundary Condition” on page 6-7](#).



- Setup:Simulation View — Select this option to setup simulation viewing parameters for: Circuit Voltages, Package Voltages, Currents and 3D(spatial) animations. For detailed information, please refer to [“Choosing the Viewing Parameters” on page 6-8](#).
- Setup:Simulation View:Delete Illegal Observation Lines — Select this option to delete invalid viewing parameters which may exist after the objects no longer exist in the project file.
- Setup:Distribution — A spatial animation of peak or average voltage values may be viewed during the simulation (refer to [“Spatial Distribution Window” on page 3-33](#)) and (if selected) statistical calculations are applied to the simulation data to derive the distribution of voltage amplitude. This data is further plotted to create the Probability Distribution curve (refer to [“2D Curves Window” on page 3-13](#)). For detailed information, please refer to [“The Distribution Dialog Box:” on page 6-18](#).

- Setup:Ext Ckt Solver — Select this option if a SPEED2000 compatible\* external circuit solver is available.



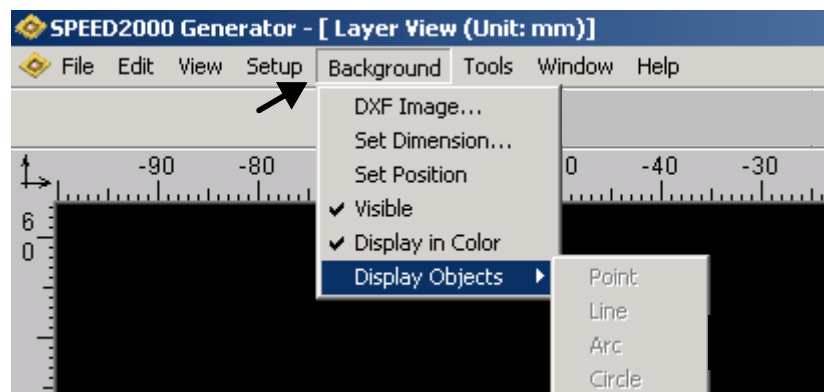
### NOTE:

\*Please contact your Sigrity sales representative for additional information.

- Setup:Linkage — Once you have built and defined circuit components the next step is to make connections between circuit nodes and package nodes — to make such linkages, open the Linkage dialog box. For detailed instructions, please refer to [“Linking Partial Circuits to the Package” on page 5-9](#).
- Setup:Edge Radiation — Select this option to record the voltages along the edges of each pair of metal planes during the simulation. For detailed instructions, refer to [“Selecting Traces and Edges for Radiation Calculation” on page 8-1](#).

### Background Menu

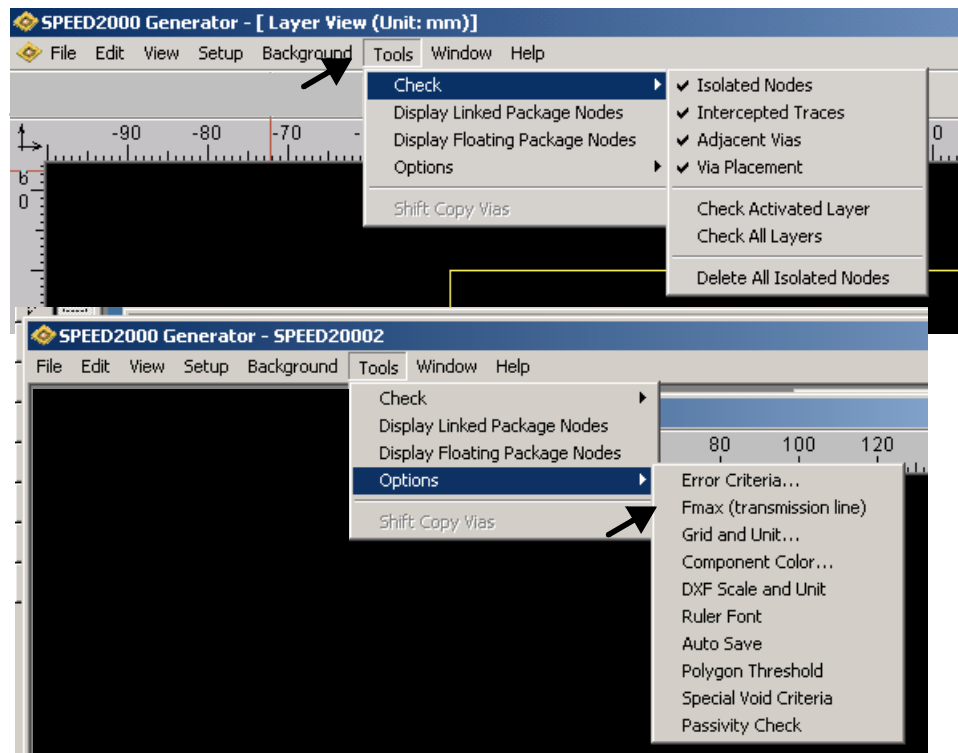
Clicking Background from the Layer View menu bar unfolds the Background drop-down menu.



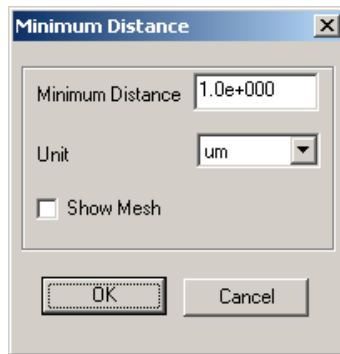
- Background:DXF Image — Select this option to load DXF files into SPDGEN. For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#)
- Background:Set Dimension — For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#)
- Background:Set Position — For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#)
- Background:Visible — For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#)
- Background:Display in Color — For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#)
- Background:Display Objects — For detailed instructions, refer to [“Loading a DXF File for the First Time” on page 4-109.](#) The display objects option further distinguishes to: Point, Line, Arc, Circle and Polyline.

### Tools Menu

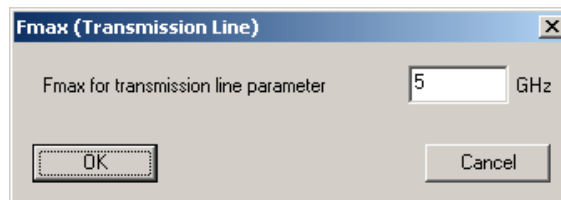
Clicking Tools from the Layer View menu bar unfolds the Tools drop-down menu.



- Tools:Check — Select this option to check the package for drawing errors that may have been introduced. For detailed information, please refer to [“Checking Drawing Errors” on page 4-99](#).
- Tools:Display Linked Package Nodes — Package nodes define specific locations in the package. They typically define the ends of traces and the upper and lower ends of vias. Package nodes can either be linked to a circuit node or floating (not linked to a circuit node). Select this option to highlight linked package nodes. For more information refer to [“Working with Circuits” on page 5-1](#).
- Tools:Display Floating Package Nodes — Package nodes define specific locations in the package. They typically define the ends of traces and the upper and lower ends of vias. Package nodes can either be linked to a circuit node or floating (not linked to a circuit node). Select this option to highlight floating package nodes. For more information refer to [“Working with Circuits” on page 5-1](#).
- Tools:Options>Error Criteria — The minimum distance value is used by the *adjacent node* and *via* error checking routines. For more information on these types of errors refer to [“Checking Drawing Errors” on page 4-99](#).

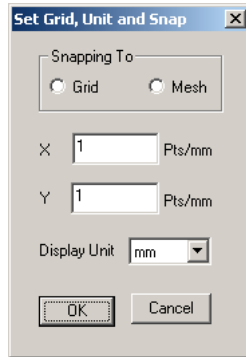


- Tools:Options:Fmax (transmission line) — For more information refer to [“Establishing Transient Simulation Parameters” on page 6-5](#).

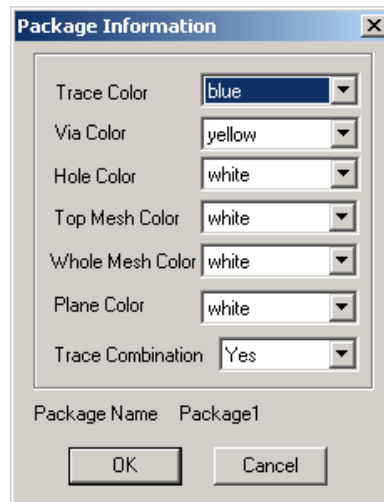




- Tools:Options:Grid and Unit — The “snap to” drawing feature is by default ‘off’. Select this option to turn on “snap to” drawing feature and establish the drawing grid units. Learn more about the mesh, grid and snap to features of Sigrity products here: [“Establishing the numerical mesh” on page 6-2](#) and [“Setting Up the Display, Drawing and Mesh Environment” on page 4-22](#).

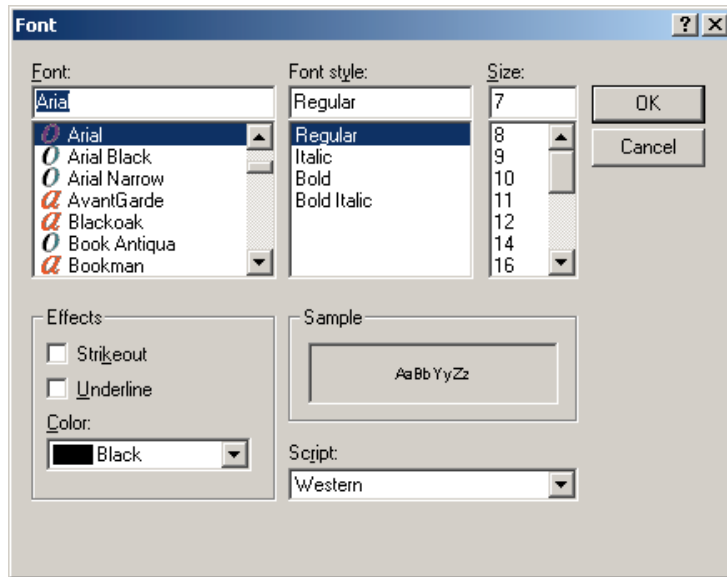


- Tools:Options:Component Color — Select this option to conveniently establish the drawing colors. For more information refer to [“To establish/change the drawing colors:” on page 4-13](#).



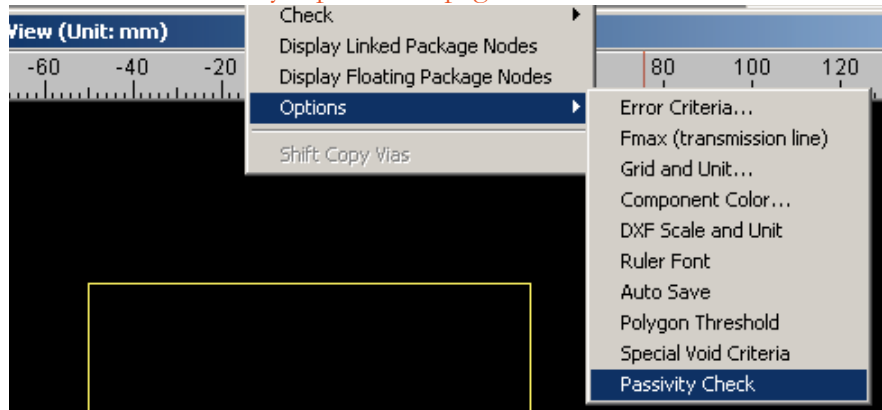
- Tools:Options:DXF Scale and Unit — Select this option to preset the DXF scale and unit. For more information refer to [“Changing Scale and Unit before Loading a DXF File” on page 4-112](#).

- Tools:Options:Ruler Font — Select this option to establish the default Font, its size and font style.



- Tools:Options:Auto Save — The SPD file will be automatically backed up in SPDGEN every 10 minutes. The name of the “back up” file will be your SPD filename + .auto extension; it will be saved within the same directory as your SPD file. It is deleted when the .spd file is saved properly. For information on how to change the automatic backup time, refer to [“Auto Save” on page 4-3](#).
- Tools:Options:Polygon Threshold — Select this option to decrease simulation time by smoothening the sharp edges of polygons. For more information refer to [“Polygon Smoothening/Simplifying” on page 4-31](#).
- Tools:Options:Special Void Criteria — Select this option to establish Special Void size criteria. For more information refer to [“Working on Shapes” on page 4-29](#).

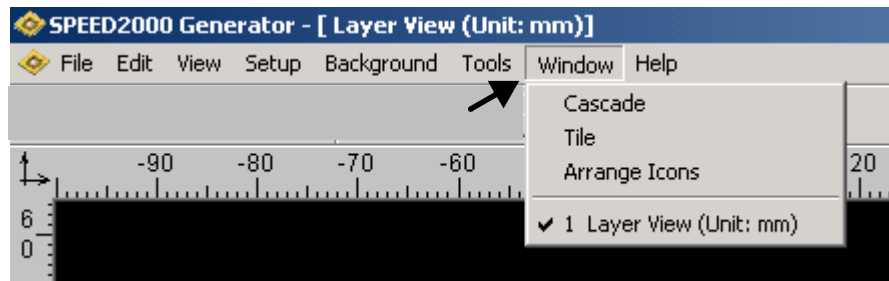
- Tools:Options:Passivity Check — For more information refer to [“Passivity Option” on page 6-7](#).



- Tools:Shift Copy Vias — For more information refer to [“Working on Vias” on page 4-68](#).

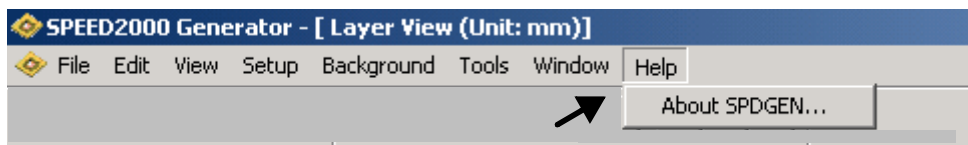
### Window Menu

Clicking Window from the Layer View menu bar unfolds the Window drop-down menu. The Window drop-down menu contains the following options: Cascade, Tile, Arrange Icons and a listing of the current Layer View windows that are open.



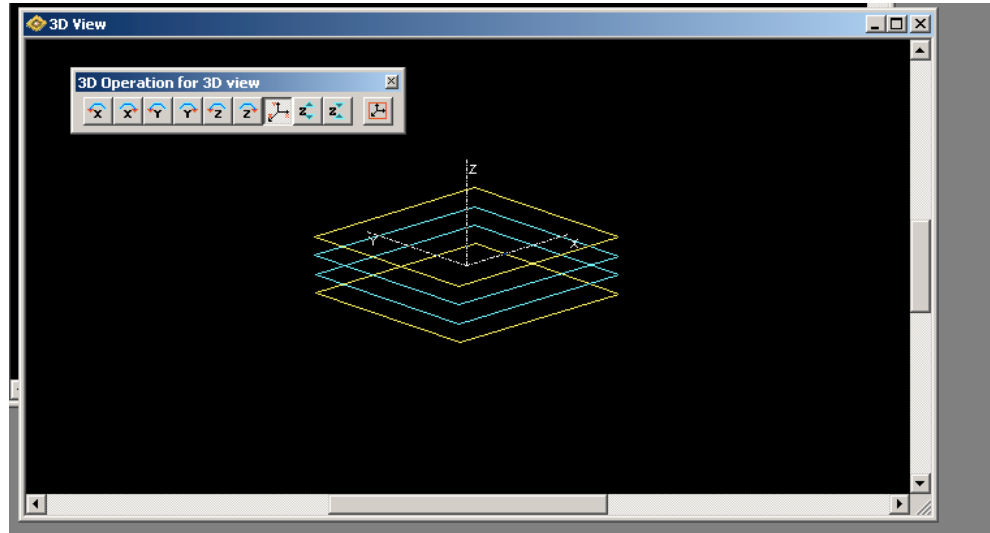
### Help Menu

Clicking Help from the Layer View menu bar unfolds the Help drop-down menu. The Help drop-down menu contains the following option: About SPDGEN. A detailed discussed of the About SPDGEN option is provided in an earlier section, please refer to [“Help Menu” on page 2-5](#).



### ***The 3D View Window***

SPDGEN enables the user to graphically work with package components in the Layer View or 3D View Windows. Selecting File:NEW or File:OPEN invokes the Layer View window display of a package. The 3D View Window can only be invoked after opening the Layer View window — from the Layer View window click View:3D View.



The 3D View Window is associated with several drop-down menus. The win-

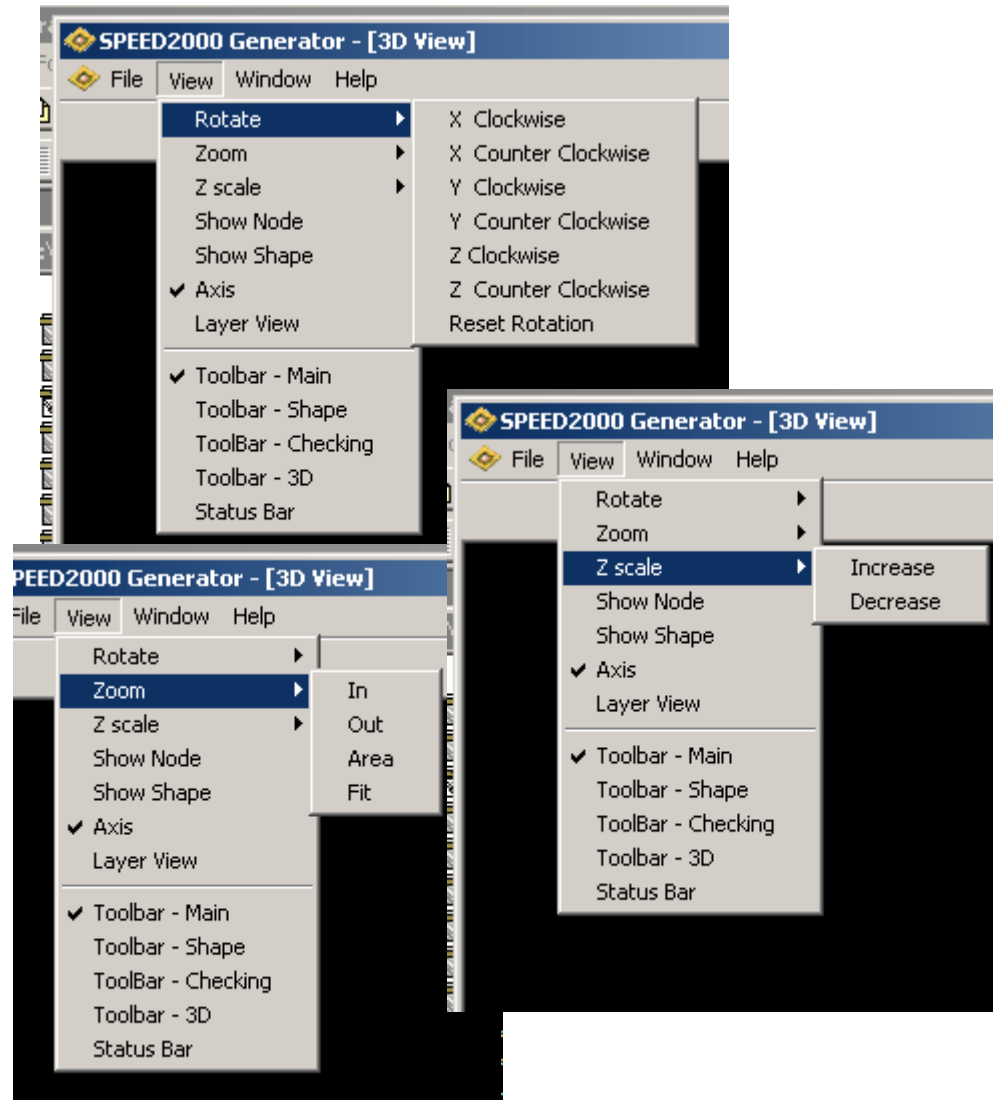
#### **NOTE:**

The 3D View window must be active to access the 3D View menu options.

dow must be active to access the 3D View menu options. We discuss the unique options available on the View (3D) drop-down menu in this section. Other drop down menus are previously discussed and their function is identical, Refer to [“ICON” on page 2-8](#), [“File Menu \(after package loaded\)” on page 2-9](#), [“Window Menu” on page 2-23](#), [“Help Menu” on page 2-5](#), [“The Toolbars of SPDGEN” on page 2-26](#).

**View Menu (3D View)**

Clicking View from the 3D View menu bar unfolds the unique 3D View drop-down menu (3D version).

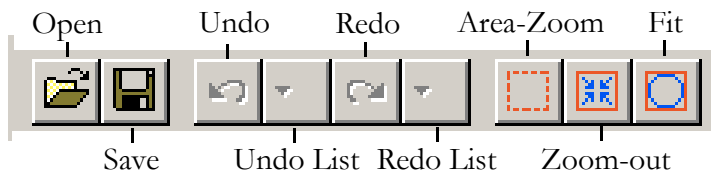


### The Toolbars of SPDGEN

This section discusses the toolbars of SPDGEN.

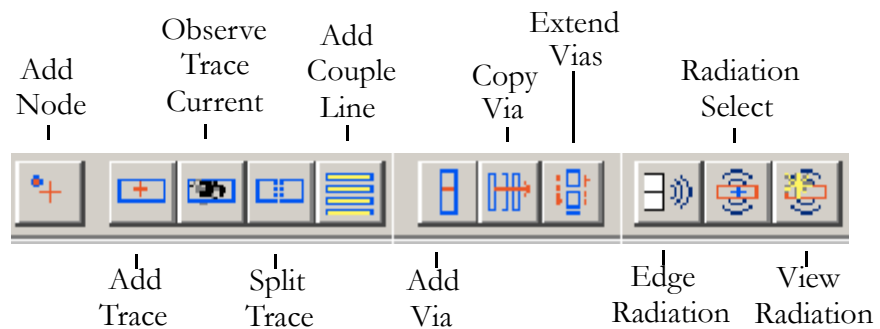
#### Toolbar - Main

This convenient tool bar has options for: Open, Save, Undo, Undo List, Redo, Redo List, Area-Zoom, Zoom-out and Fit. The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the main toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the View: *Toolbar* options.



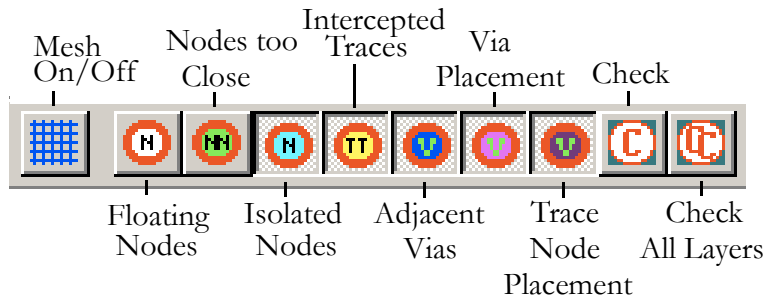
#### Toolbar - Object

This convenient tool bar has options for: Add Node, Add Trace, Observe Trace Current, Split Trace, Add Couple Line, Add Via, Copy Via, Extend Vias, Edge Radiation, Radiation Select and View Radiation. The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Object toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the View: *Toolbar* options.



**Toolbar - Checking**

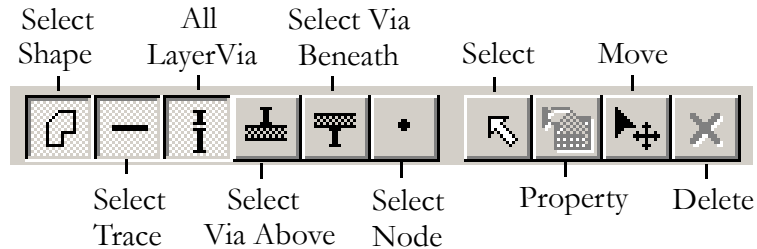
This convenient tool bar has options for:



Show Mesh, Floating Nodes, Node to Close, Isolated Nodes, Intercept Trace, Adjacent Vias, Via Placement, Trace Node Placement, Check and Check All Layers. For more information, refer to [“Checking Drawing Errors” on page 4-99](#). The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Checking toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the *View:Toolbar* options.

**Toolbar - Select**

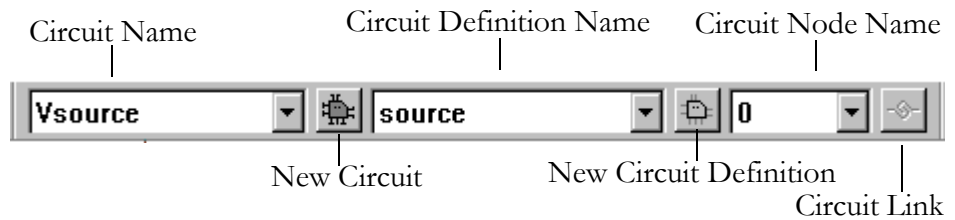
This convenient tool bar has options for:



Select Shape, Select Trace, Select Entire Via, Select Via Above, Select Via Beneath, Select Node, Select, Property, Move and Delete. For more information, refer to [“Selecting Objects” on page 4-21](#). The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Select toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the *View:Toolbar* options.

**Toolbar - Circuit**

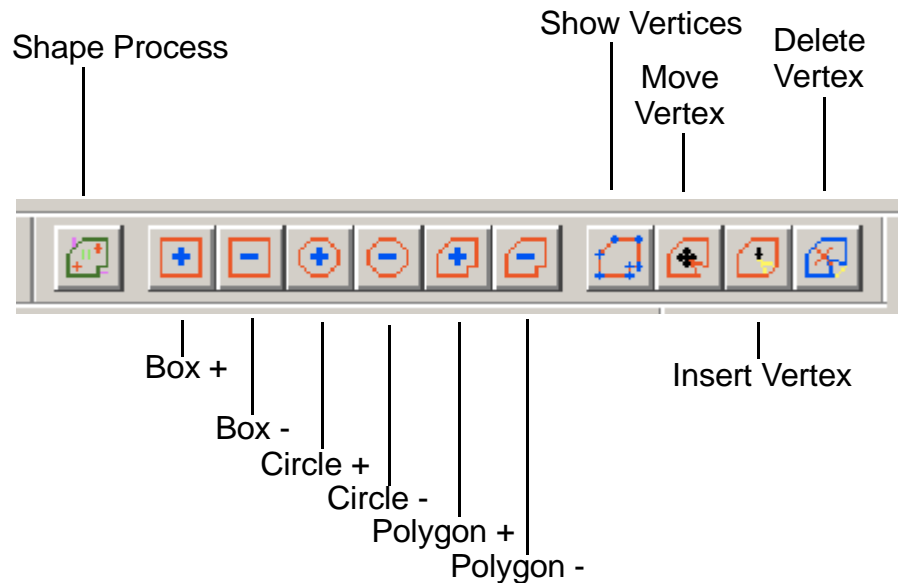
This convenient tool bar has options for:



Circuit Name, New Circuit, Circuit Definition Name, New Circuit Definition, Circuit Node Name and Circuit Link. For more information, refer to [“Working with Circuits” on page 5-1](#). The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Circuit toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the *View:Toolbar* options.

**Toolbar - Shape**

This convenient tool bar has options for:

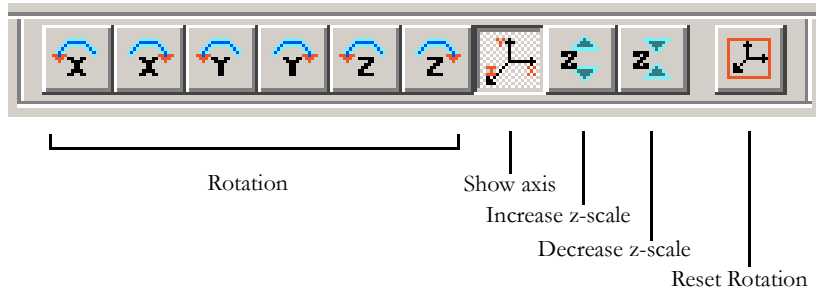


For more information, refer to [“Working on Shapes” on page 4-29](#). The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Circuit toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the *View:Toolbar* options.



**Toolbar - 3D**

The SPDGEN:Toolbar-3D toolbar contains convenient options for:



For more information, refer to [“The 3D View Window” on page 2-24](#). The toolbar can be stowed in the window panel just beneath the window menu bar area or left-click in any non-button area of the toolbar, hold and drag and the Circuit toolbar becomes a “floating” toolbar. A floating toolbar can be closed using the toolbar upper-right close box which appears. Closed toolbars can be viewed using the View: *Toolbar* options.



## SPDSIM Features Overview

SPEED2000 is composed of two executable modules SPDGEN and SPDSIM. This chapter presents an overview of the features, menus and dialog boxes of the SPDSIM module. An overview of the features, menus and dialog boxes of the SPDGEN module is presented in [“SPDGEN Features Overview” on page 2-1](#). This chapter presents an answer to the questions: What are the features of SPDSIM? What is the feature for? Where is it located? and How do I invoke it? A procedural, step-by-step, presentation of *how to use* the features of SPDSIM is available in the following chapters. We provide pointers to specific portions of the procedural descriptions through-out this features overview presentation.

### Menu Option Citation

SPDSIM uses drop-down menus to facilitate User interaction. The menus available change depending upon the Window that is activated. In this chapter, menu options are cited in conjunction with their parent option, the main menu title and the activated window title as a path delimited with colons (:) — SPEED2000 module:window title:menu title:menu option. For example:

### SPDSIM:2d Curves:View:Navigator

Where:

“SPDSIM” is the *SPEED2000 module* (SPEED2000 is composed of SPDGEN and SPDSIM executable modules),

“2D Curves” is the *window title* (SPDSIM has 5 windows: Radiation, Spatial Distribution, 2D Curves, Layer Display and Package Display),

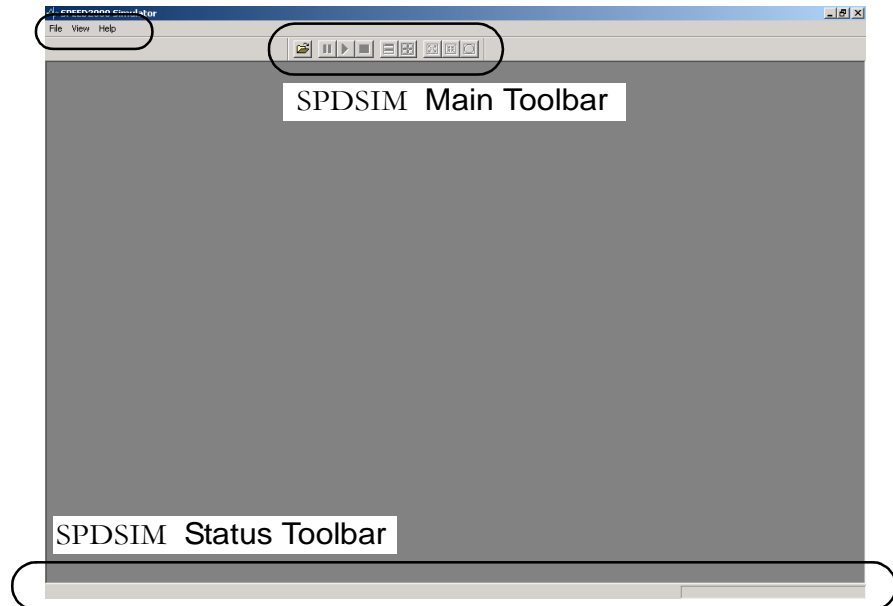
“View” is the drop-down *menu title*

and “Navigator” is the *menu option*.

### Initial Main Menu Bar

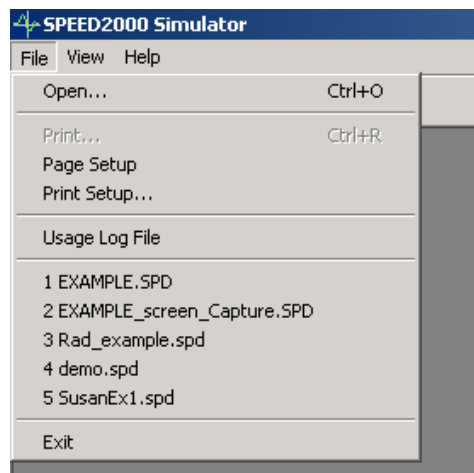
In its initial state SPDSIM appears as a Graphical User Interface (GUI) work space with three menu bar options (File, View and Help), the SPDSIM Main Toolbar and the status bar.

#### SPDSIM Main Menu bar

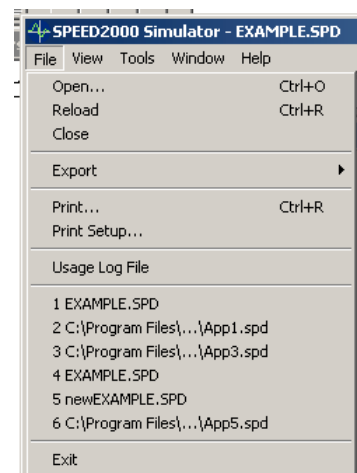


#### File Menu (before package loaded)

Clicking File from the main menu of SPDSIM unfolds the File drop-down menu with options unique to the status "before package loaded". The options that appear on the File drop-down menu are different "after the package is loaded". For more information refer to "[File Menu \(after package loaded\)](#)" on [page 3-8](#).



**before package loaded**

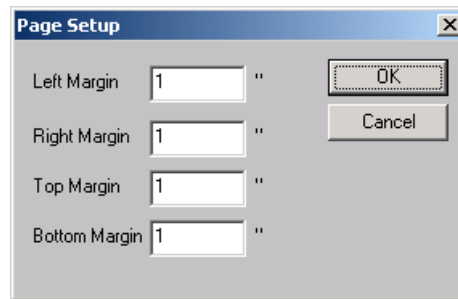


**after package loaded**

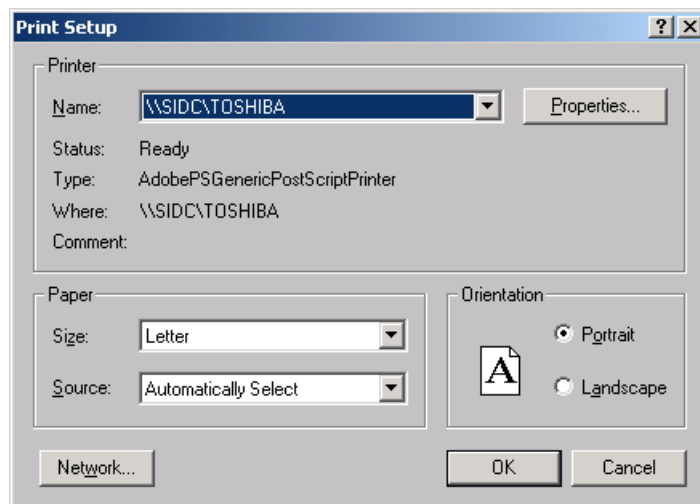
*Two "File" Menus*

The File drop-down menu contains the following options:

- SPDSIM:File:Open — Selecting this option invokes the Windows OPEN dialog box, where you can browse (at the system level) to select any SPEED format file (.spd). Refer to the [“The Package View Windows” on page 3-7](#) for information related to opening a SPEED file (.spd), the system messages you may see and many other user interface details after a package is successfully opened.
- SPDSIM:Print — This option is not available until the work space contains a printable object.
- SPDSIM:Page Setup — Selecting this option displays the Page Setup dialog box that permits the user to enter Margin values that will be applied when printing any Curve files (.cur) from SPDSIM.

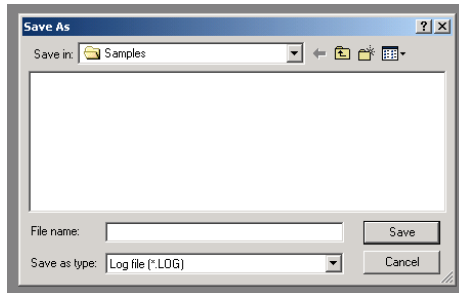


- SPDSIM:File:Print Setup — Selecting this option invokes the Windows Print Setup dialog box, where you can select the target printer and print job attributes.



- SPDSIM:File:Usage Log File — Selecting this option will invoke the Save As dialog box, which permits you to enter the name of a usage log file. The log file maintains an administrative records of the usage of the SPDSIM

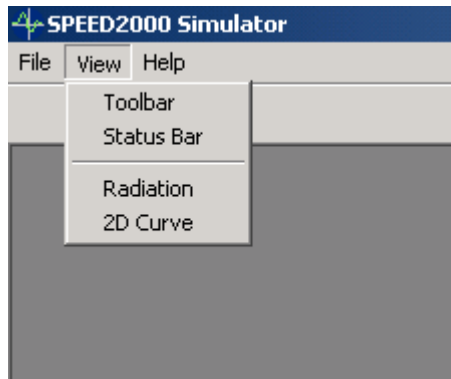
application — it records how many times the application was invoked, by what user on what system. This information may be useful to management to help decide if additional copies of SPEED2000 are warranted.



- SPDSIM:File:Recent File Listing — Listing of the last four SPEED files accessed.
- SPDSIM:File:Exit — Selecting this option closes any open SPEED files and the SPDSIM application.

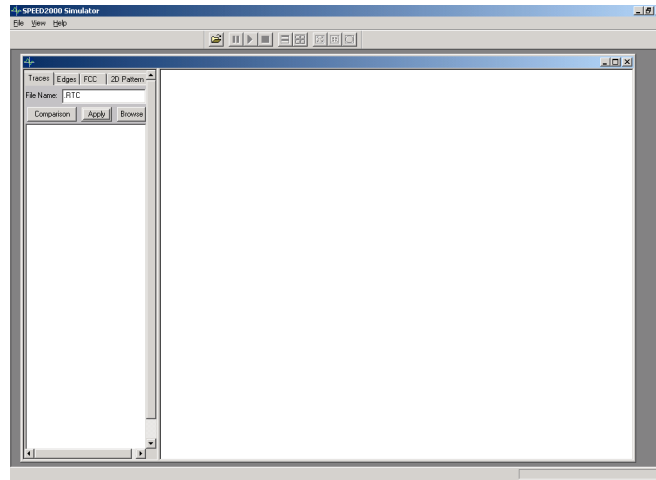
### View Menu

Clicking View from the main menu of SPDSIM unfolds the View drop-down menu. The View option drop-down menu contains the following options:

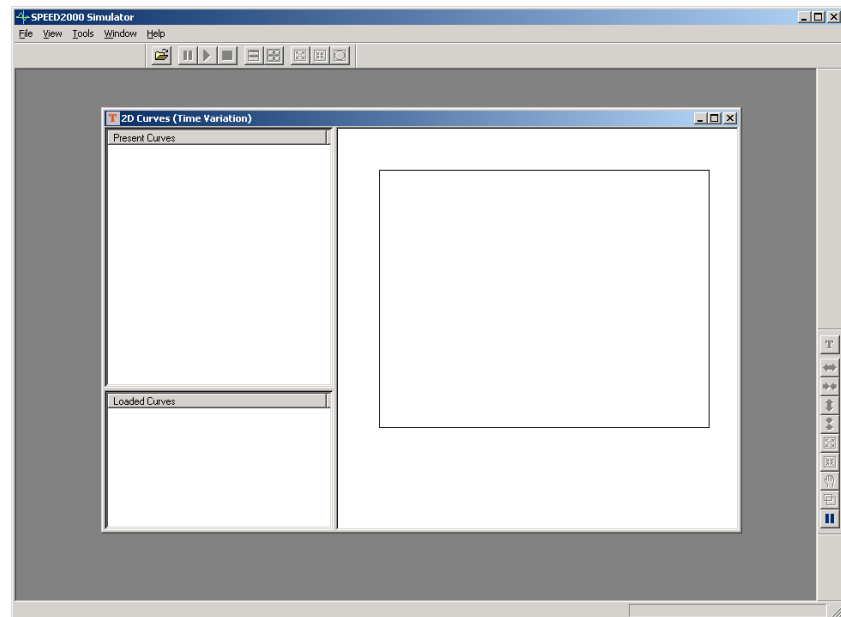


- SPDSIM:View:Toolbar — This option provides no functionality at present.
- SPDSIM:View:Status Bar — Toggling this option will display and close the SPDSIM status bar at the bottom of the window. For more information on the Status Bar, refer to: [“Status Bar” on page 3-6](#).
- SPDSIM:View:Radiation — Selecting this option will display the radiation application graphical user interface (GUI). Radiation calculation is a post-processing feature which enables users to simulate the far field radiation based on the voltages and currents modeled in traces and planes during the

transient simulation. Refer to [“Calculating Radiation” on page 8-1](#) for more information on How To use the SPDSIM radiation feature.

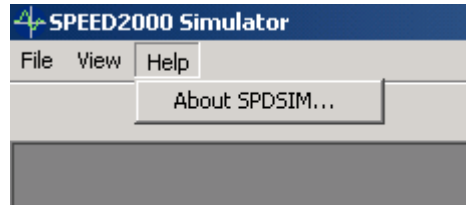


- SPDSIM:View:2D Curve — Selecting this option displays an unpopulated 2D Curves window. Displaying the window enables you to load previously generated SPEED 2D Curve files (.cur). Note that the menu options available change when the 2D Curves window is displayed. You may then choose to print or export any displayed Curve file (.cur) as bitmap files (.bmp). Refer to [“2D Curves Window” on page 3-13](#) for more information.

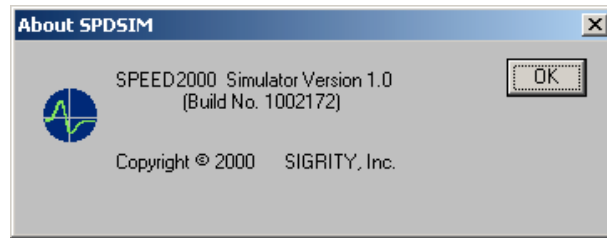


### Help Menu

Clicking Help from the main menu of SPDSIM unfolds the Help drop-down menu. The Help option drop-down menu contains the following option:



- SPDSIM:Help:About SPDSIM — Selecting this option will display the About SPDSIM dialog box which details the version and build specific information of the SPDSIM module. If you are anticipating contacting Sigrity Technical Support, you may find it helpful to display this dialog box and record the version and build information.



### Toolbar - Main (before a package is loaded)

After a package is loaded the SPDSIM:Main toolbar provides convenient access to several functions. In its initial status, before a package is loaded, only the Open function is active.



Press the Open button to open a SPEED (.spd) format file package for simulation.

### Status Bar

The status bar, at the bottom of the application window, is used to display system task status messages, the purpose of a button or menu option and (when the work space windows contain a package) the simulation time step information. The number of time steps of the simulation are entered using SPDGEN.



For more information on entering time steps see [“Establishing Computation Parameters” on page 6-2](#).

#### SPDSIM Status Bar



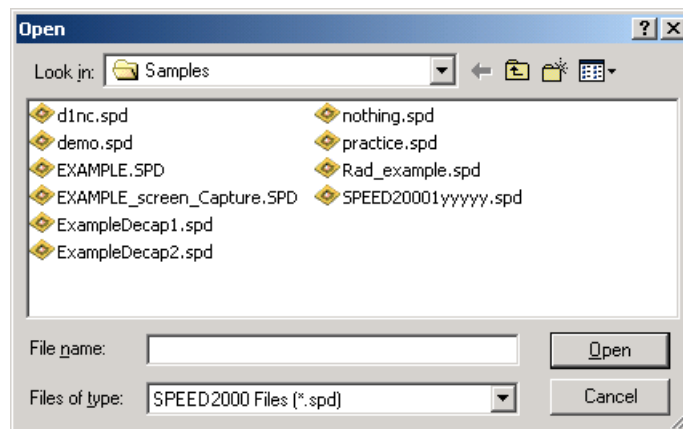
Left side of the Status Bar:  
system task status messages  
and/or the purpose of  
a button or menu option



Right side of the Status Bar:  
displays the TimeSteps of  
the simulation

### The Package View Windows

Selecting SPDSIM:File:OPEN displays the Open dialog box. SPDSIM will attempt to open and process files with the .spd extension.



#### NOTE:

SPEED files (.spd) must have been processed by SPDGEN at least once before entering SPDSIM. Simulation and Viewing parameters need to be specified using SPDGEN. For more information refer to [“Establishing Simulation Parameters” on page 6-1](#).

As SPDSIM processes the SPEED (.spd) package file the user may see several progress messages (below). The purpose of these messages is to provide the User with feedback regarding the progress of the package verification.

- Running...
- Reading Package ...

- Doing Package Arrangement ...
- PtCkctRearrange ...

### Error Log File

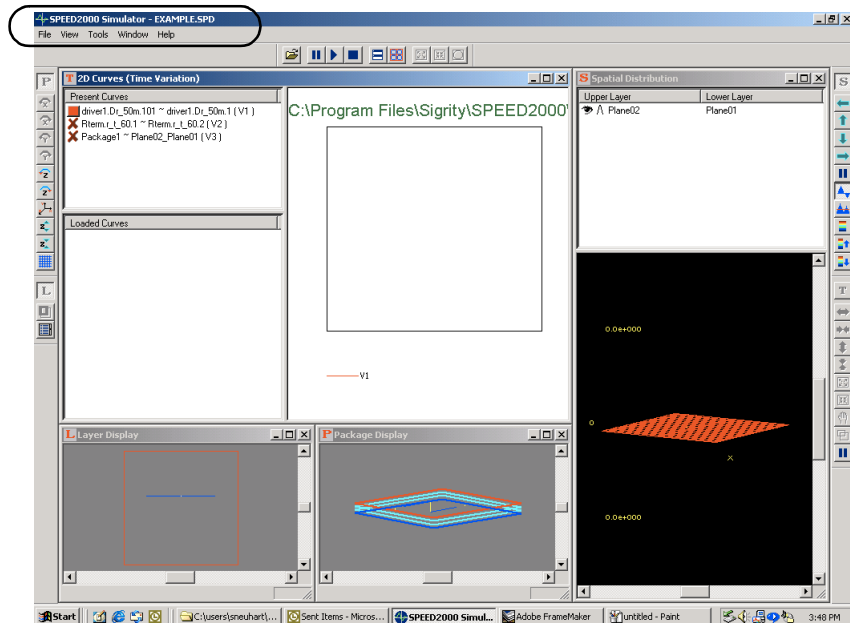
After the preprocessing checks are completed successfully, SPDSIM will display four package-related windows: 2D Curves (Time Variation), Spatial Distribution, Layer Display and Package Display. If the .spd file is incomplete, or it has errors detectable by SPDSIM, it will fail to open the file and prompt you to check the error log of the problematic .spd file. The error log file produced will have the same file name as the SPD file, but with the xxxxxxxx\_SPDSIM.err suffix. Where xxxxxx equals the input .spd file name.

### NOTE:

If all of the package view windows are not displayed, select Window (drop-down menu), then select To Default option.

### Common Window Menus

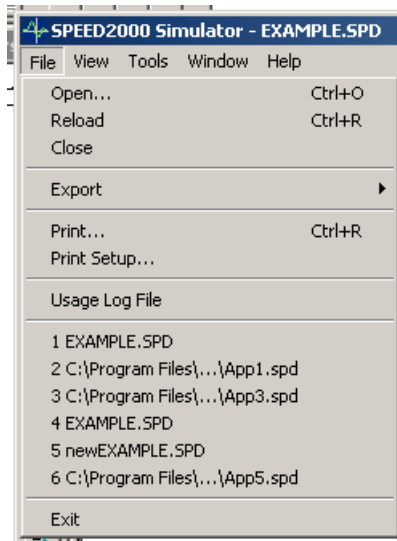
Let's begin by discussing the functionality and features common to all SPDSIM windows (after a package is open): File, Window and Help.



### File Menu (after package loaded)

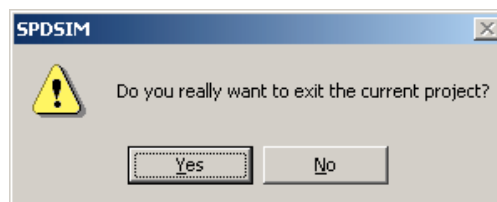
Clicking File from the SPDSIM menu bar after a package has been loaded unfolds the “after package loaded” File drop-down menu. For more informa-

tion on the File menu “before the package is loaded” refer to [“File Menu \(before package loaded\)” on page 3-2](#).



The File drop-down menu (after a package is loaded) contains the following options:

- SPDSIM:File:Open — Selecting this option displays the Windows OPEN dialog box where you can browse (at the system level) to select any SPEED format file (.spd). After you select a file and press Open, a message box to confirm if you want to exit the current project file is displayed. If you answer yes, the currently open package file is closed and the file selected is opened.
- SPDSIM:File:Reload — Selecting this option reloads the opened .spd file.
- SPDSIM:File:Close — Selecting this option displays a message box to confirm if you want to exit the current project file. This is referencing the currently open package file. If you select Yes, the opened package (.spd) file will be closed and the SPDSIM application will remain open. The position

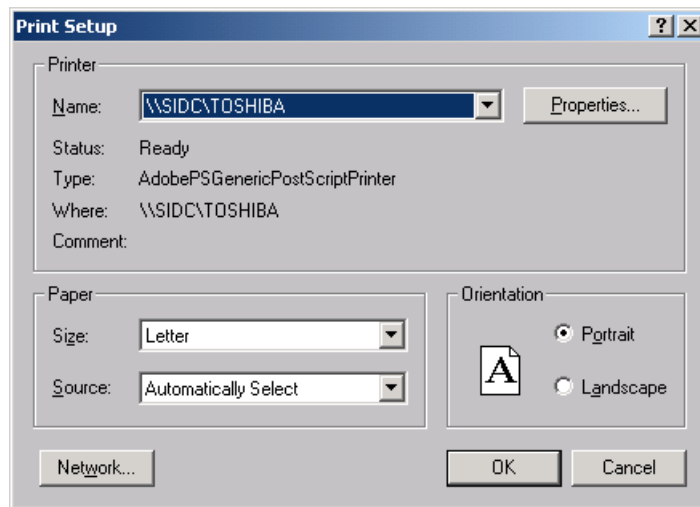


of the four SPDSIM windows is saved when closed. The next time SPDSIM opens that package file (the .spd file) the same window layout would be shown.

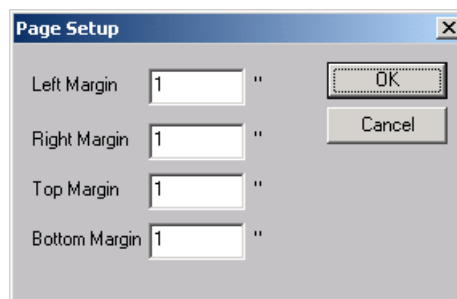
- SPDSIM:File:Export — After a simulation has been run, selecting this option permits the User to save the 2D Curves window display as a bitmap

image file (.bmp). This option is only available in conjunction with the 2D Curves window contents.

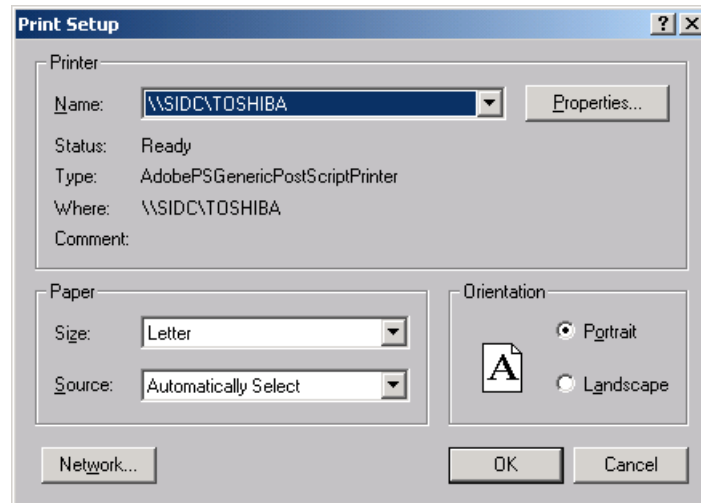
- SPDSIM:Print — This option is not available until the work space contains a printable object. When the work space contains a package design, displayed components of each window can be printed. That is, Curve files (.cur) can be printed from the 2D Curves window, animation snapshots can be printed from the Spatial Distribution window, top-down views of each package layer can be printed from the Layer Display window and dimensional views of the package components can be printed from the Package Display window.
- SPDSIM:File:Print Setup — Selecting this option invokes the Windows Print Setup dialog box, where you can select the target printer and print job attributes.



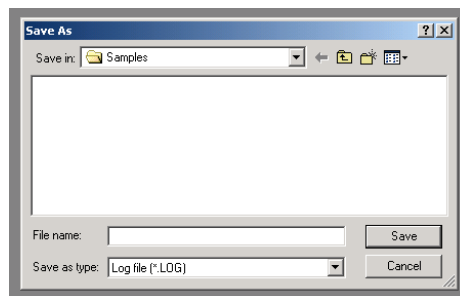
- SPDSIM:Page Setup — Selecting this option displays the Page Setup dialog box that permits the user to enter margin values that will be applied when printing any Curve files (.cur) from SPDSIM.



- SPDSIM:File:Print Setup — Selecting this option invokes the Windows Print Setup dialog box, where you can select the target printer and print job attributes.



- SPDSIM:File:Usage Log File — Selecting this option will invoke the Save As dialog box, which permits you to enter the name of a usage log file. The log file maintains an administrative records of the usage of the SPDSIM application — it records how many times the application was invoked, by what user on what system. This information may be useful to management to help decide if additional copies of SPEED2000 are warranted.



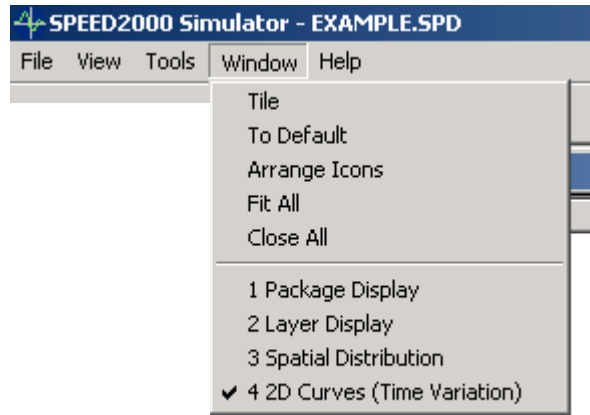
- SPDSIM:File:Recent File Listing — Listing of the last four SPEED files accessed.
- SPDSIM:File:Exit — Selecting this option closes any open SPEED files and the SPDSIM application.

### Help Menu

The Help drop-down menu was discussed previously. The options and behaviors are identical here — please refer to the beginning of this chapter: [“Help Menu” on page 3-6.](#)

**Window Menu**

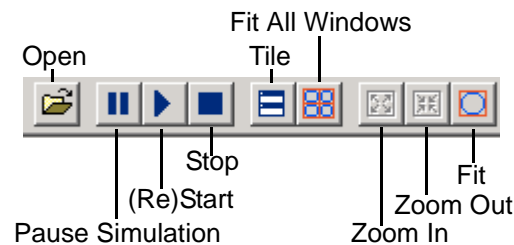
Clicking Window from the SPDSIM menu bar unfolds the Window drop-down menu. The Window drop-down menu controls the overall workspace window display — it contains the following options:



- SPDSIM:Window:Tile — Select this option to conveniently reorganize the package windows that are open.
- SPDSIM:Window:To Default — Select this option to open all package windows.
- SPDSIM:Window:Arrange Icons — Select this option to conveniently arrange any minimized package windows at the bottom of the workspace.
- SPDSIM:Window:Fit All — Select this option to fit the package components displayed on the Layer Display, Spatial Distribution and Package Display window.
- SPDSIM:Window:Close All — Select this option to close all package windows and exit the package.
- Package Window List — Select any file listed to open a related window that may be closed.

**Toolbar - Main (after the package is loaded)**

The SPDSIM:Main toolbar provides convenient access to the following options. Note that all of the buttons of the toolbar are active after a package is open.



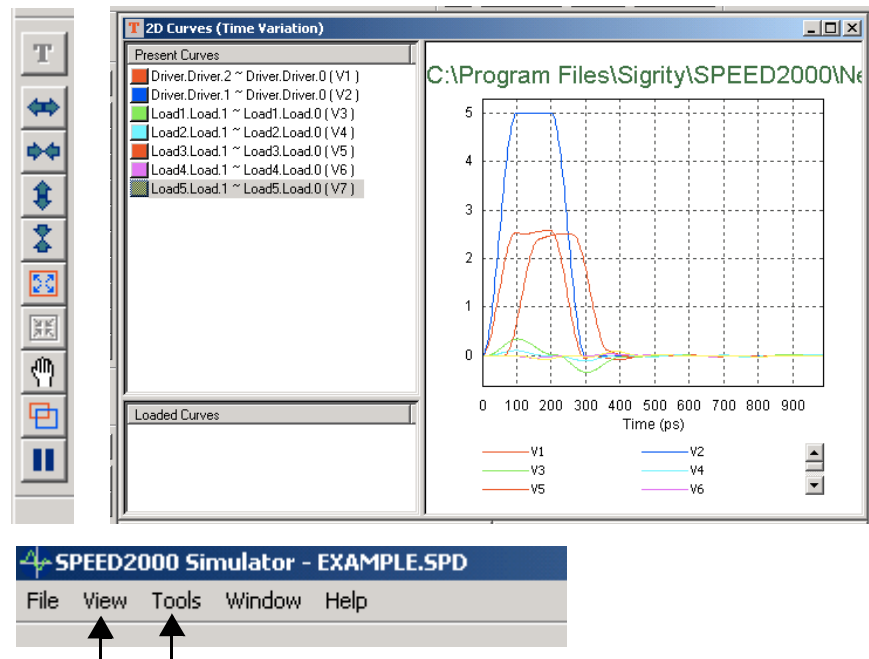
- Open — Press this button to open a SPEED (.spd) format file package for simulation.
- Pause Simulation — Press this button to pause a simulation that is in progress.
- (Re)Start — Press this button to start or RE-start a simulation that has been paused or stopped.
- Stop — Press this button to stop a simulation that is in progress.
- Tile — Press this button to conveniently resize and display the package windows that are open. Minimized or closed windows will not be affected.
- Fit All Windows — Press this button to conveniently fit all of the open windows in the application work space.
- Zoom In — Press this button to Zoom In on the Layer Display, Package Display and Spatial Distribution windows.
- Zoom Out — Press this button to Zoom Out on the Layer Display, Package Display and Spatial Distribution windows.
- Fit — Press this button to fit the components displayed on the Layer Display, Package Display and Spatial Distribution windows.

### **2D Curves Window**

The SPDSIM 2D Curves window displays the transient simulation data that results from the SPDSIM transient simulation of package components previously selected (using SPDGEN) and it can also be used to view previous simulation data (saved as .cur files). The 2D Curves window is the user interface to access post processing tools for 2D data analysis and viewing — 2D analysis tools are: Time Domain display of voltage and current simulation data, Frequency Domain display of simulation data and Probability plot of voltage distribution between planes. The title bar of the 2D Curves window tracks the analysis tool in use. The 2D Curves window has three panels:

- Present Curves — this panel provides the user interface to select among the present curves displayed. The curve results displayed are the result of the present simulation. Recall, the display of curves is determined by choices and options made using SPDGEN. For more information, refer to [“Choosing the Viewing Parameters” on page 6-8.](#)
- Loaded Curves — this panel provides the user interface to load curve files (.cur) from previous simulations — so the user may conveniently compare results obtained earlier.
- 2D Animation area — this panel displays a 2D animation of the potential difference between any two circuit nodes or any adjacent package planes.

Curves displayed in this panel are selected from the Present Curves panel and the Loaded Curves panel.



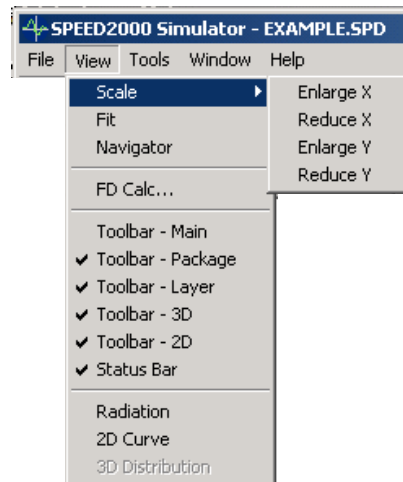
The 2D Curves Window has two unique drop-down menus: View and Tools

The 2D Curves window has a dedicated toolbar and two unique drop down menus, see [“View Menu \(2D Curves window\)” on page 3-15](#) and [“Tools Menu” on page 3-17](#). The File, Window and Help drop down menus contain options and behave as discussed in previous sections: [“File Menu \(before package loaded\)” on page 3-2](#), [“Window Menu” on page 3-12](#) and [“Help Menu” on page 3-6](#).SPEED2000 Data File Flow



**View Menu (2D Curves window)**

- The SPDSIM:2D Curves:View menu has the following options:

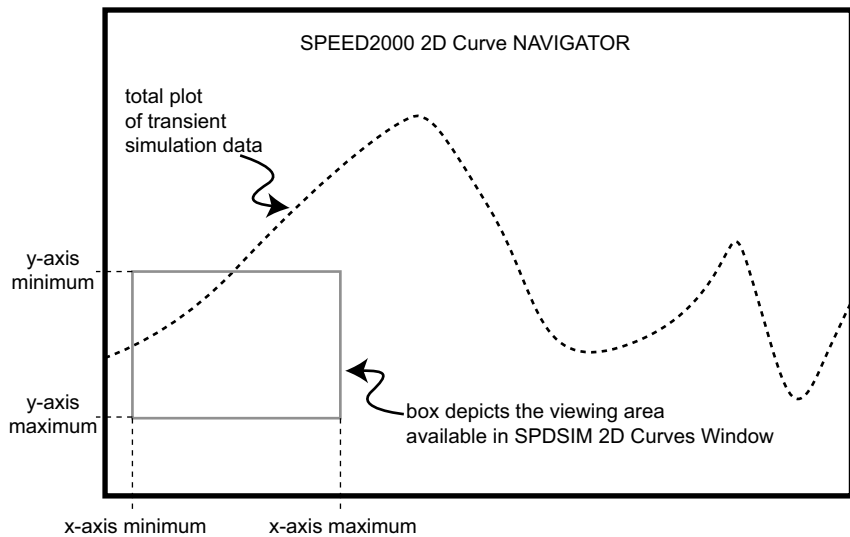
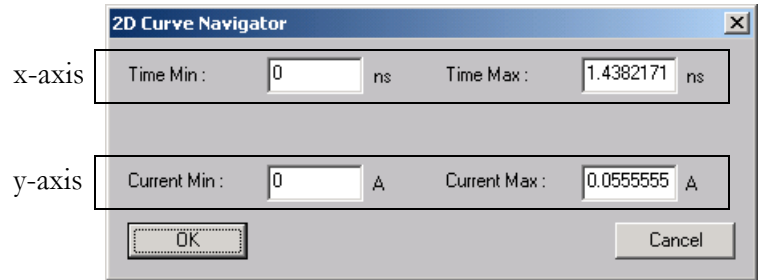
**NOTE:**

Some View menu options are activated only after the package simulation has been run.

- SPDSIM:2D Curves:View:Scale — This menu option invokes an additional fly-out menu which enables the User to enlarge or reduce the X and Y-axis of the 2D plot.
- SPDSIM:2D Curves:View:Fit — Select this option to fit the display of the plotted information in the 2D Curves window.
- SPDSIM:2D Curves:View:Navigator — Select this option to precisely define the displayed area of a 2D plot. Enter the x-axis and y-axis minimum and maximum values to define a box that circumscribes the area of interest within the entire plotted area (see figure below). The Navigator is context sensitive for all of the 2D curve plots available. For example, the y-axis window information is constrained by Current minimum and Current maximum for the current related plot; the y-axis window information is

constrained by Voltage minimum and Voltage maximum for the voltage related plot.

The axis are context sensitive for information displayed:



- SPDSIM:2D Curves:View:FD Calc... — Select this option to display a Fourier Calculator popup dialog box. SPDSIM provides this calculation tool to enable operations for two curves or variables in the frequency domain. For more information, refer to [“Obtaining Frequency Domain Parameters” on page 7-13.](#)
- SPDSIM:2D Curves:View:Toolbar -Main — Select this option to display the SPDSIM Main toolbar. Refer to: [“Toolbar - Main \(before a package is loaded\)” on page 3-6.](#)
- SPDSIM:2D Curves:View:Toolbar - Package — Select this option to display the SPDSIM Package Display toolbar. Refer to: [“Package Display Toolbar” on page 3-43.](#)
- SPDSIM:2D Curves:View:Toolbar - Layer — Select this option to display the SPDSIM Layer Display toolbar. Refer to: [“Layer Display Toolbar” on page 3-41.](#)

- SPDSIM:2D Curves:View:Toolbar - 3D — Select this option to display the SPDSIM Spatial Distribution toolbar. Refer to: [“Spatial Distribution Toolbar” on page 3-36.](#)
- SPDSIM:2D Curves:View:Toolbar - 2D — Select this option to display the SPDSIM 2D Curves toolbar. Refer to: [“2D Curves Toolbar” on page 3-23.](#)
- SPDSIM:2D Curves:View:Status Bar. — Select this option to show and hide the application status bar. For more information on the Status Bar refer to: [“Status Bar” on page 3-6.](#)
- SPDSIM:2D Curves:View:Radiation — Select this option to display the Radiation post processing user interface. For more information refer to: [“Calculating Radiation” on page 8-1.](#)
- SPDSIM:2D Curves:View:2D Curve — Selecting this option invokes another 2D Curves window. The additional 2D Curves window permits the user to load a previously generated SPEED curve file (.cur) such that a convenient comparison of previous simulation results are possible.

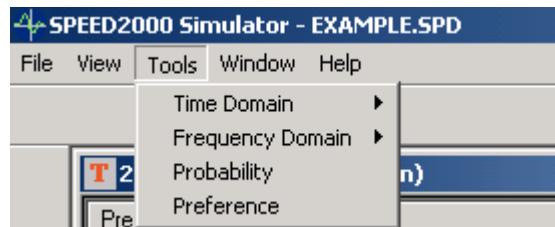
**NOTE:**

The new 2D Curves window is initially displayed on top of the existing 2D Curves window. Use the pointer to move and reposition the new window as necessary for convenient comparison of curve data. Click the close button to close the 2D Curves window.

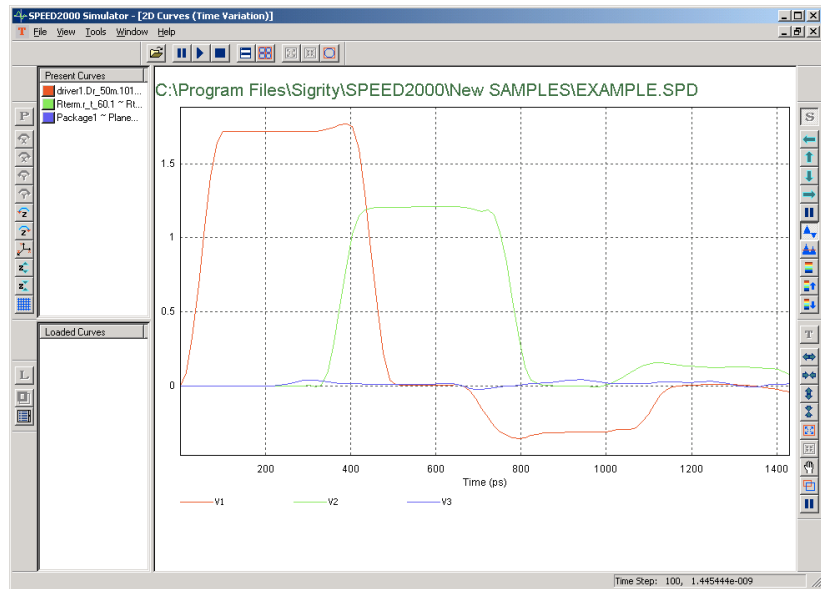
- and SPDSIM:2D Curves:View:3D Distribution. When the Spatial Distribution window is closed, this option is active. Selecting it will open the Spatial Distribution window. For more information refer to: [“Spatial Distribution Window” on page 3-33.](#)

**Tools Menu**

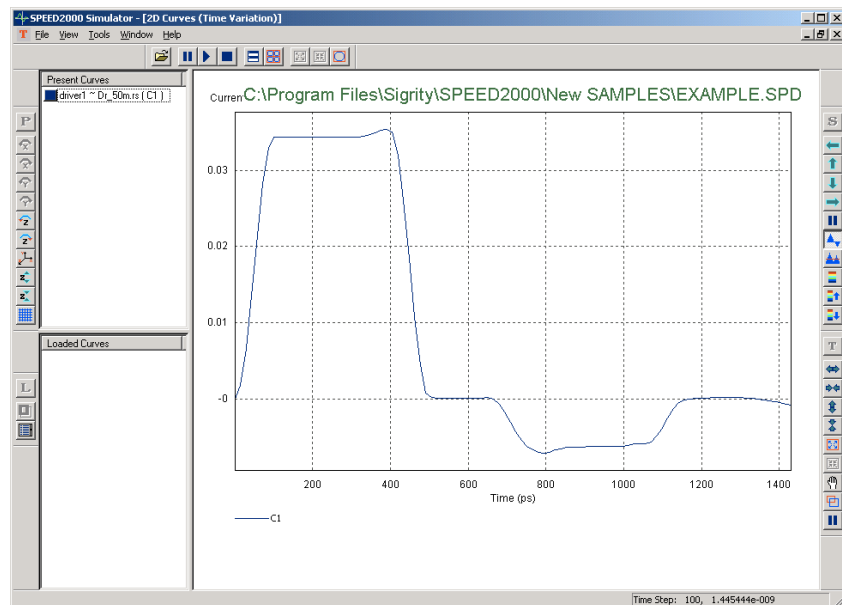
The SPDSIM:2D Curves:Tools menu contains eight total options: Time Domain (Voltage and Current), Frequency Domain (a Fourier series of amplitude, Phase Shift, Real Part and Imaginary Part), Voltage Probability plot and a Preferences dialog box. We overview each option in this section.



- SPDSIM:2D Curves:Tools:Time Domain: The Time Domain option expands to Voltage and Current.
  - SPDSIM:2D Curves:Tools:Time Domain:Voltage — Selecting this option displays a plot of the voltage waveform versus time. Recall, circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8.](#)



- SPDSIM:2D Curves:Tools:Time Domain:Current — Selecting this



option displays a plot of current waveform versus time. Recall, cir-

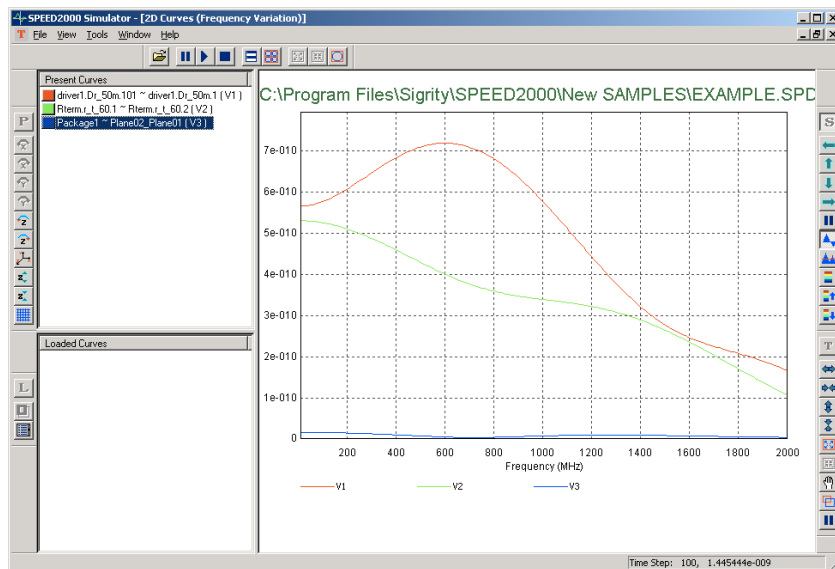
circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8.](#)

- SPDSIM:2D Curves:Tools:Frequency Domain — SPDSIM provides a Fourier transform function to convert 2D plots of simulation data from time domain to frequency domain. In frequency domain, the variable values are complex numbers. A complex number contains a real part and an imaginary part. It can also be presented by its amplitude and phase. Select this option to display a fly-out menu with additional options for Amplitude, Phase, Real Part and Imaginary Part.

#### NOTE:

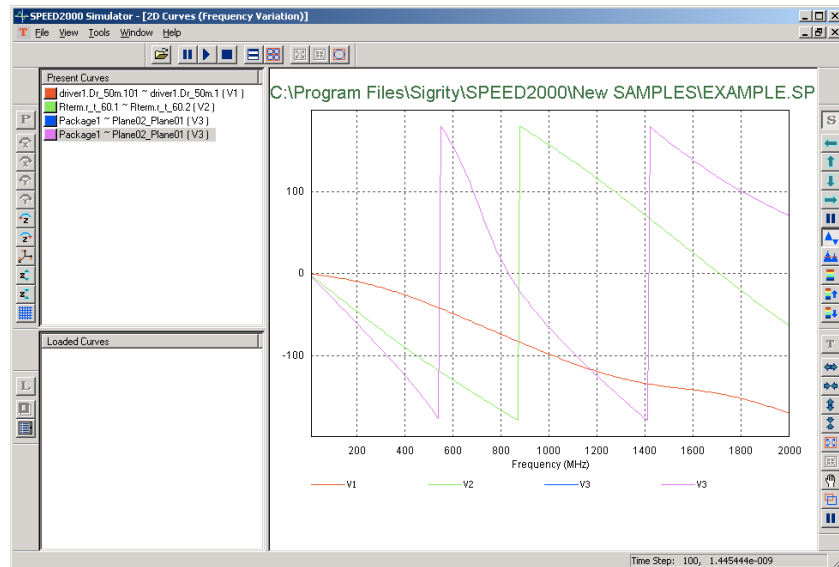
During the simulation, the 2D Curves window permits the user to view the voltages or currents in time domain. Other viewing capabilities (available on the Tools menu) are provided as post-process functions and can only be launched when the transient simulation stops or it is paused.

- Frequency Domain:Amplitude — Selecting this option displays a plot



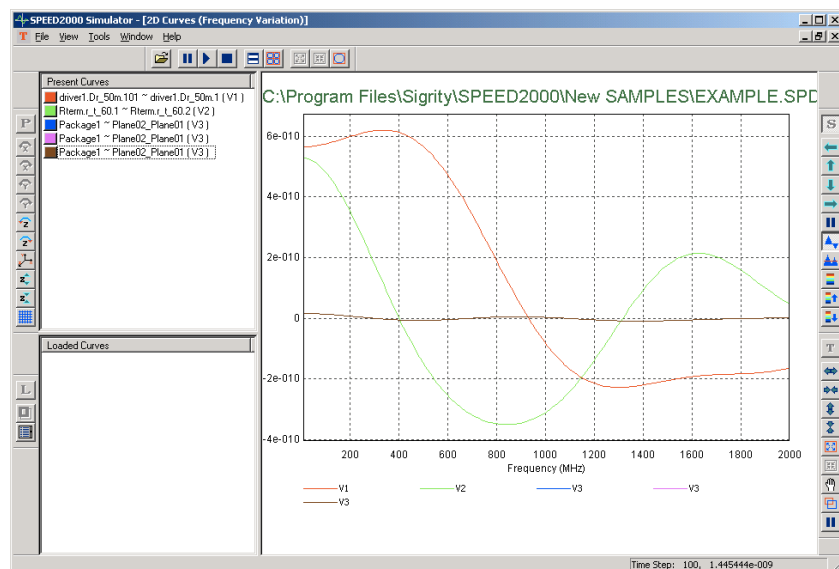
of amplitude of selected variables displayed in the 2D Current (Time Domain) window. Recall, circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8.](#)

- Frequency Domain:Phase — Selecting this option displays a plot of



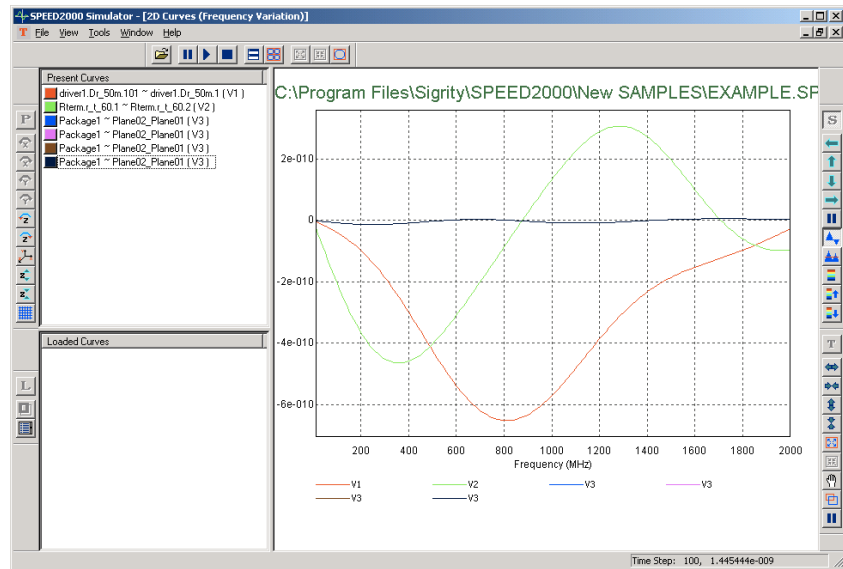
Phase (in degrees) of selected variables. Recall, circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8.](#)

- Frequency Domain:Real Part — Selecting this option displays a plot of



of REAL part of selected variables. Recall, circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8.](#)

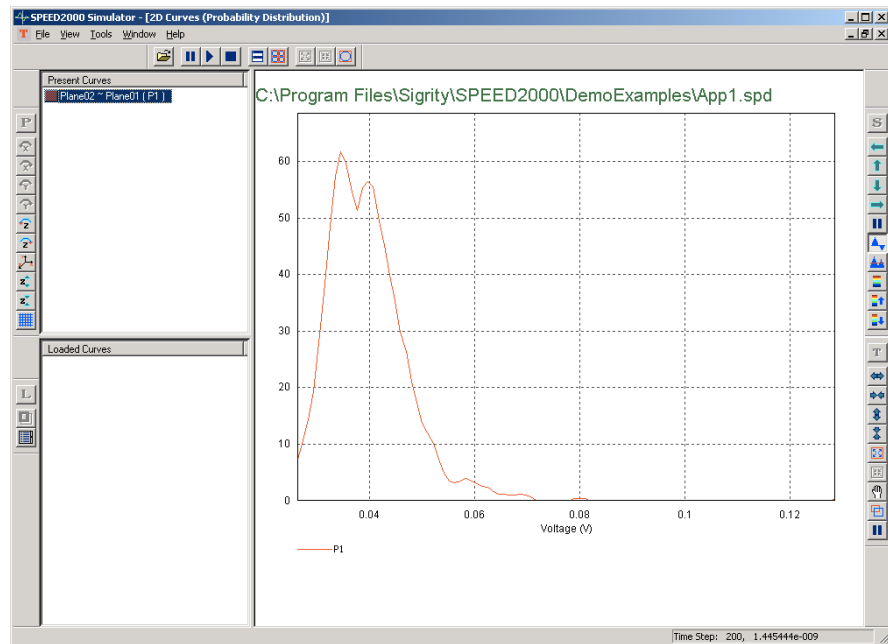
- Frequency Domain:Imaginary Part — Selecting this option displays a



plot of IMAGINARYpart of selected variables. Recall, circuits are selected for simulation using SPDGEN. For more information refer to [“Choosing the Viewing Parameters” on page 6-8](#).

- SPDSIM:2D Curves:Tools:Probability — Selecting this option displays a Probability Distribution plot of voltage between planes. For more information refer to [“Choosing Parameters for Voltage Viewing and Analysis” on page 6-18](#). Two parameters can be adjusted with respect to the display of

the Probability Distribution plot: nBars and Factor. Access these parameters using the Preference dialog box — discussed next..



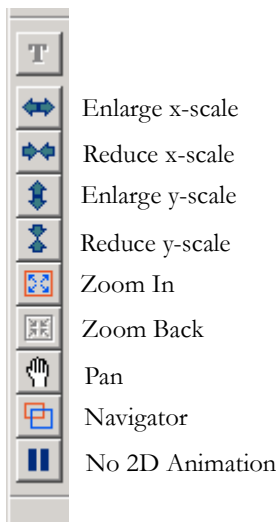
- SPDSIM:2D Curves:Tools:Preference — Selecting this option displays a dialog box where the user can enter preferences related to the Probability Distribution plot and the Fourier Transformation tools.

- Statistics Curve:nBars — Enter a value, 1 to 500, to affect the overall display of the data points along the x-axis. For example, a value of 10 in a voltage range of 0 to 1, would result in data points at: .1, .2,



.3, .4, .5, .6, .7, .8, .9 and 1. A value of 100 in the same voltage range would result in data points at .01, .02, .03 ... .99 and 1.

- **Statistics Curve:Factor** — Enter a value, 1 to 21, to affect the smoothness of the plot. Specifically, if the smoothing factor is N, then the displayed probability distribution curve is smoothed from the original curve so that the value at any point on the smoothed curve is the average of N values at adjacent points on the un-smoothed curve.”
- **Fourier Transformation:nFreq** — Enter a value for the number of sample points of frequency between 0 to Fmax.
- **Fourier Transformation:Fmax** — Enter a value for the upper limit of frequency when making a Fourier Transform.



### 2D Curves Toolbar

The SPDSIM:2D Curves toolbar provides convenient access to:

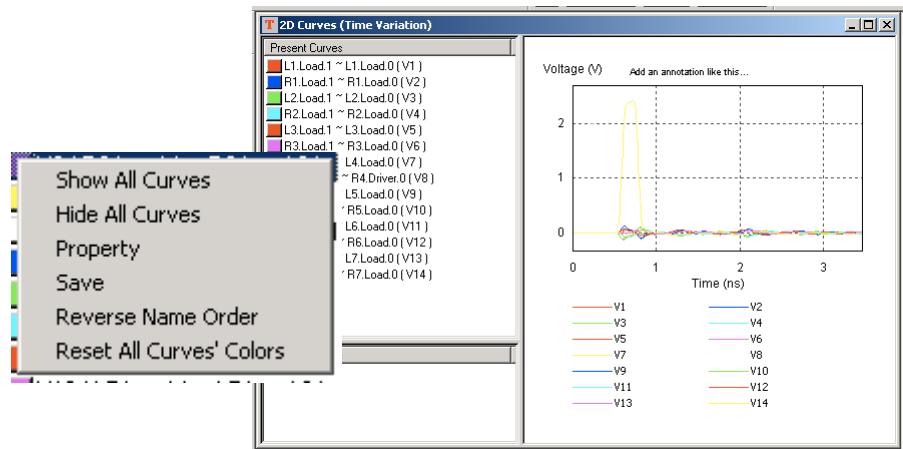
- **Enlarge x-scale** — Select this option to increase the horizontal scale of the information displayed in the 2D Curves window.
- **Reduce x-scale** — Select this option to reduce the horizontal scale of the information displayed in the 2D Curves window.
- **Enlarge y-scale** — Select this option to increase the vertical scale of the information displayed in the 2D Curves window.
- **Reduce y-scale** — Select this option to reduce the vertical scale of the information displayed in the 2D Curves window.
- **Zoom In** — Select this option to zoom in on the information displayed in the 2D Curves window.
- **Zoom Back** — Select this option to zoom out from the information displayed in the 2D Curves window.
- **Pan** — Select this option to position the information plotted most favorably to the x and y axis. The pan option changes the pointer to a hand that appears to grab and position the plot under the window.
- **Navigator** — Select this option to display the navigator dialog box. Navigator is also an option available from the 2D Curves View Menu. Detailed information on the navigator can be found at [“View Menu \(2D Curves window\)” on page 3-15](#).
- **No 2D Animation** — Select this option to halt the real-time display of the 2D data animation in the 2D Curves window. This speeds up the transient simulation processing significantly ( often 50%~200%).

### Right-click Pop-Up Menus (context sensitive)

The 2D Curves Package Window has several *context sensitive* menus that pop-up on mouse right-click — that is, the menu that pops up is dependent upon where the mouse is positioned (in the 2D Curves window) when the user performs a right-click.

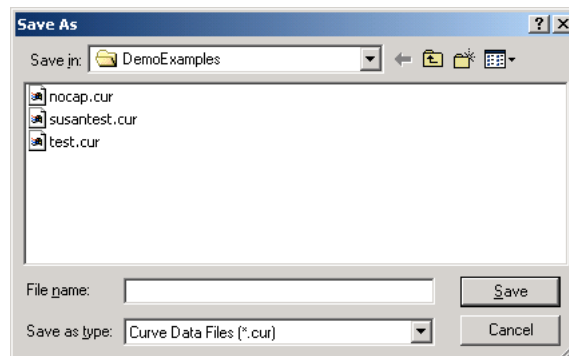
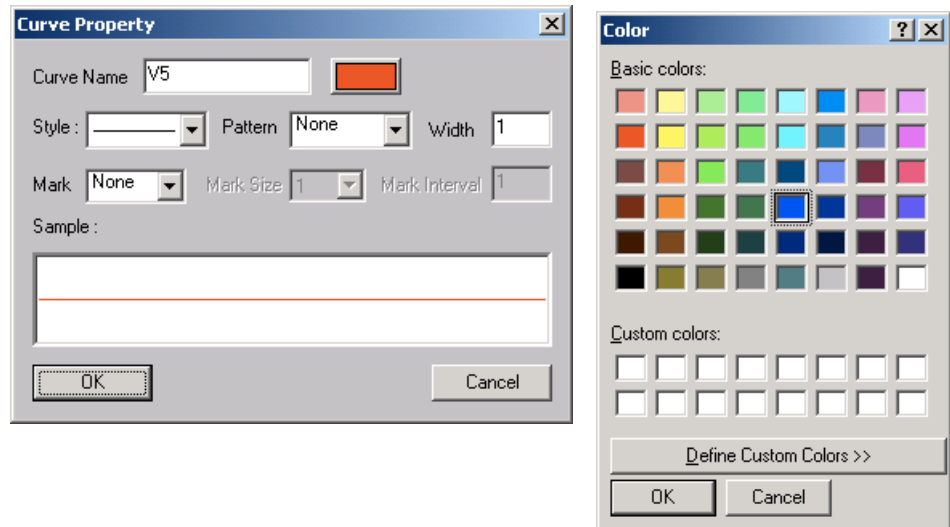
**Right-click Present Curves Menu**

Right-clicking in the Present Curves portion of the 2D Curves window invokes the Present Curves popup menu:



- **Show All Curves** — Selecting plots of simulated circuits to hide or display can be done individually, by clicking on the curve name in the Present Curves panel of the 2D Curves window, or as a group. Select ‘Show All Curves’ to display all curves plotted. Circuits are selected for simulation using SPDGEN. Refer to [“SPDGEN Features Overview” on page 2-1](#).
- **Hide All Curves** — Selecting plots of simulated circuits to hide or display can be done individually, by clicking on the curve name in the Present Curves panel of the 2D Curves window, or as a group. Select ‘Hide All Curves’ to hide all curves plotted. Circuits are selected for simulation using SPDGEN. Refer to [“SPDGEN Features Overview” on page 2-1](#).
- **Property** — Selecting this option displays the Curve Property dialog box. Within this dialog box the user can edit the display curve properties. Properties selected are displayed in the Sample area of the Curve Property dialog box.
  - **Curve Name** — enter letters, numbers and symbols to identify the curve.
  - **Curve Color** — click on the color box to display the Color pop up menu. Select any Basic color displayed or you may define a custom color, based upon the attributes of: Hue, Saturation, Luminance, Red, Green and Blue.
  - **Curve Style** — select from the line styles available on the drop-down menu.
  - **Curve Pattern** — select from the line pattern styles available on the drop-down menu.
  - **Curve Width** — enter a positive integer.

- Curve Mark — select from the styles available on the drop-down men to identify the nodes of the curve.



- Save — Select this option to display the Save As dialog box. This dialog box enables the user to save simulation data as a SPEED Curve Data File (.cur). This file can be loaded later to be used for comparison with the results from other simulations; or, it can be loaded and printed or exported to a bitmap (.bmp). Simulation data may be saved (as a .cur file) for each kind of plot file generated, that is: Time Domain - Current, Time Domain - Voltage, Proba-

bility, Frequency Domain - Amplitude, Frequency Domain - Phase Shift, Frequency Domain - Imaginary Part and Frequency Domain - Real Part.

**NOTE:**

The internal file header is unique for each type of curve file saved. You may wish to give the saved curve files descriptive names, to simplify loading them subsequently to the unique curve display environments. For example, SPDSIM will not permit a Probability type .cur file to be loaded to the Time Domain - Current display environment.

Also, if there is curve data that you don't want to save, you need to hide them before selecting save.

And finally, each voltage or current curve is assigned a default name within the saved curve data file. For example, the first voltage curve in the Curve Selection dialog box will have a name V1, whereas the currents will be assigned to C1, C2, etc. The name of a curve (with a prefix of the package name) is shown in the parenthesis after the node/element information for each curve. You may want to change the default curve name when saving the curve to a data file — to make it easier for you to distinguish the curve the next time the curve is loaded from the saved data file.

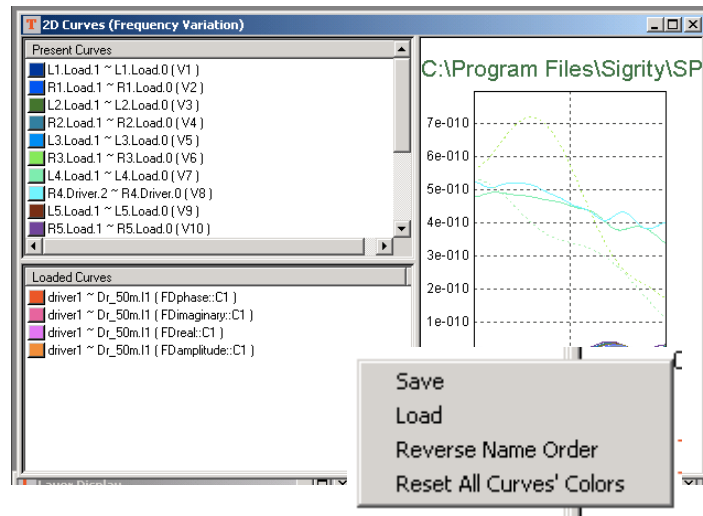
- Reverse Name Order — Select this option to reverse the circuit node identifier and user supplied name display information.
- Reset All Curves' Colors — The color of curves may easily be changed individually by right-clicking on them and selecting "property". Sometimes, because there are so many curves, it is not easy to select unique color combinations. Select this option to set a unique color for each curve modeled in the package based upon internal color algorithms.

**NOTE:**

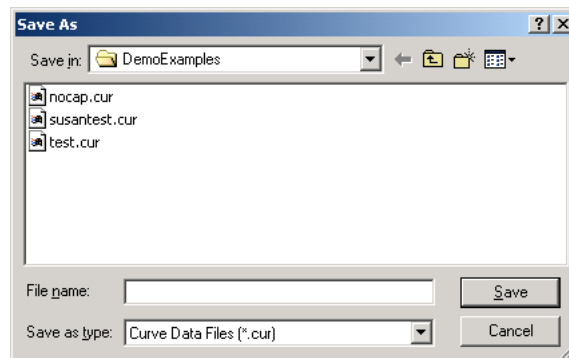
There is NO UNDO of Reset All Curves' Color option.

**Right-click Loaded Curves Menu**

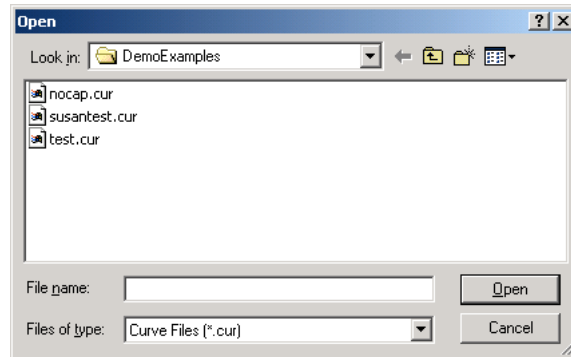
Right-clicking in the Loaded Curves portion of the 2D Curves window invokes the Loaded Curves popup dialog box:



- **Save** - Select this option to display the Save As dialog box. This dialog box enables the user to save simulation data as a SPEED Curve Data File (.cur). This file can be loaded later to be used for comparison with the results from other simulations; or, it can be loaded and printed or exported to a bitmap (.bmp). Simulation data may be saved (as a .cur file) for each kind of plot file generated, that is: Time Domain - Current, Time Domain - Voltage, Probability, Frequency Domain - Amplitude, Frequency Domain - Phase Shift, Frequency Domain - Imaginary Part and Frequency Domain - Real Part.

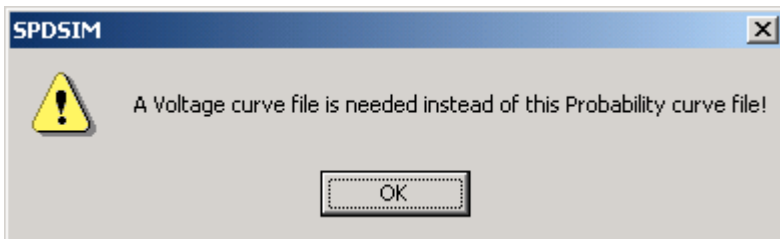


- Load — Select this option to display the Open dialog box. Plot data may be



saved (as a .cur file) for each kind of plot file generated, that is: Time Domain - Current, Time Domain - Voltage, Probability, Frequency Domain - Amplitude, Frequency Domain - Phase Shift, Frequency Domain - Imaginary Part and Frequency Domain - Real Part. Each kind of curve data file saved has a unique display environment, signified by an internal file header, that is unique for each type of curve file saved. For example, SPDSIM will not permit a Probability type .cur file to be loaded to the Time Domain - Current display environment (see below) You may wish to give the saved curve files descriptive names, to simplify loading them subsequently to the unique curve display environments.

For example, attempting to load a Probability Curve file (.cur) when the Time Domain > Voltage tool is selected, will cause this message to appear.



- Reverse Name Order — Select this option to reverse the circuit node identifier and user supplied name display information.
- Reset All Curves' Colors — The color of circuits may easily be changed individually by right-clicking on them and selecting "property". Select this

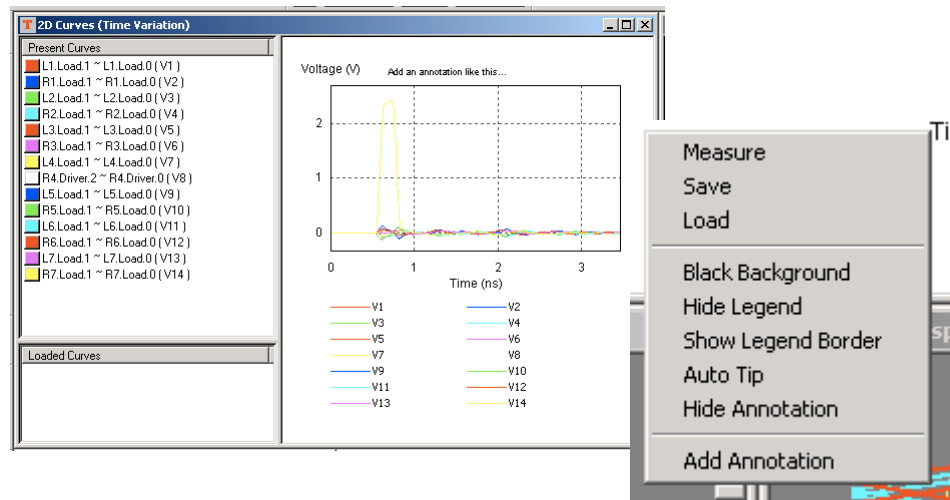
option to set a unique color for each circuit modeled in the package based upon internal algorithms.

#### NOTE:

There is NO UNDO of Reset All Curves' Color option.

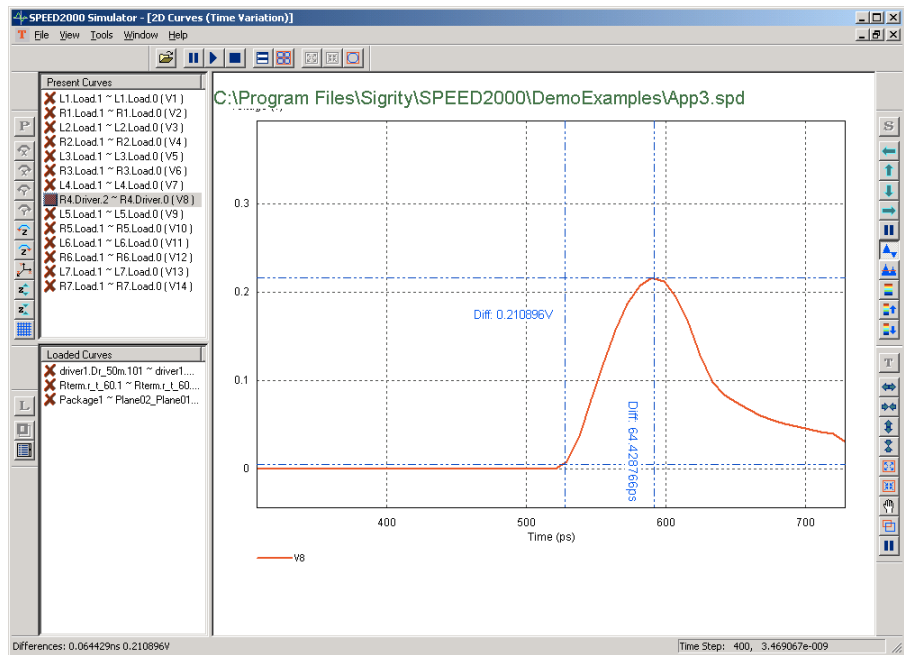
#### Right-click Plotted Information Menu (Time Domain)

Right-clicking in the plotted information portion of the 2D Curves window invokes the Plotted Information popup menu:

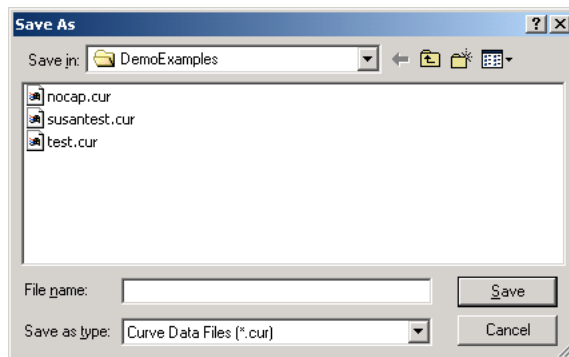


- **Measure** — Select this option to display a context sensitive measurement feature which displays the value differences of the horizontal and vertical axis guides. The guides can be set to circumscribe a specific area of a curve to aid observation and analysis. For example, in the screen capture below,

the guides have been set to reveal that the curve area of interest has a .21V rise in a 64.4ps period.

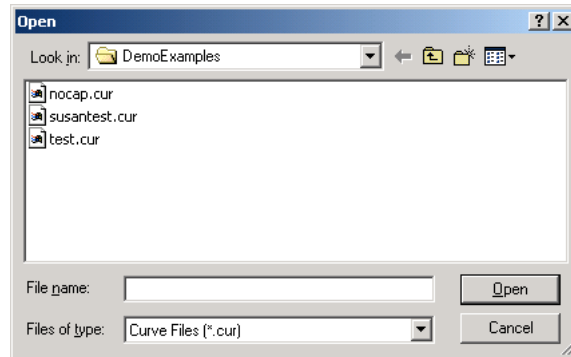


- **Save** - Select this option to display the Save As dialog box. This dialog box enables the user to save simulation data as a SPEED Curve Data File (.cur). This file can be loaded later to be used for comparison with the results from other simulations; or, it can be loaded and printed or exported to a bitmap (.bmp). Simulation data may be saved (as a .cur file) for each kind of plot file generated, that is: Time Domain - Current, Time Domain - Voltage, Probability, Frequency Domain - Amplitude, Frequency Domain - Phase Shift, Frequency Domain - Imaginary Part and Frequency Domain - Real Part.





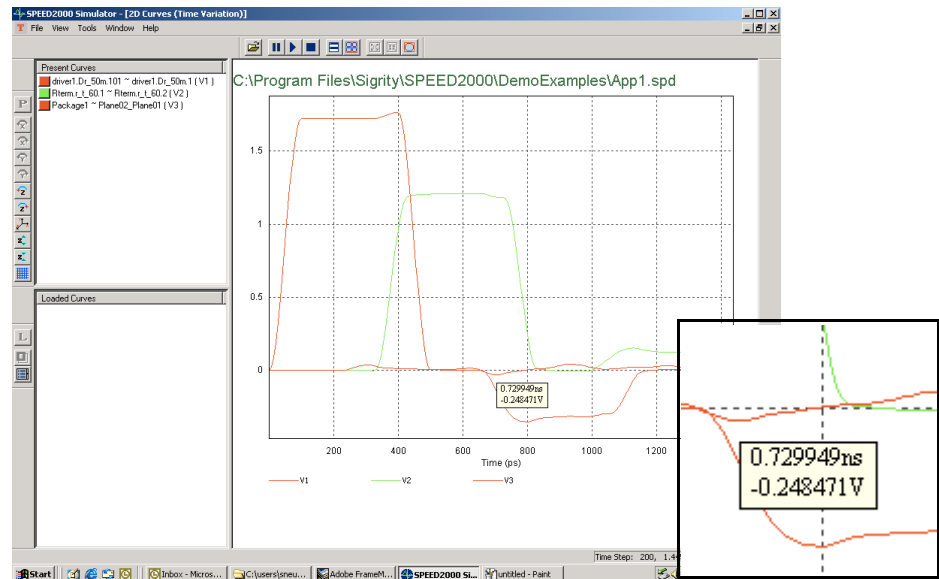
- Load — Select this option to display the Open dialog box. Plot data may be



saved (as a .cur file) for each kind of plot file generated, that is: Time Domain - Current, Time Domain - Voltage, Probability, Frequency Domain - Amplitude, Frequency Domain - Phase Shift, Frequency Domain - Imaginary Part and Frequency Domain - Real Part. Each kind of curve data file saved has a unique display environment, signified by an internal file header, that is unique for each type of curve file saved. For example, SPDSIM will not permit a Probability type .cur file to be loaded to the Time Domain - Current display environment. You may wish to give the saved curve files descriptive names, to simplify loading them subsequently to the unique curve display environments. Refer to [“SPEED2000 Data File Flow” on page 7-1](#).

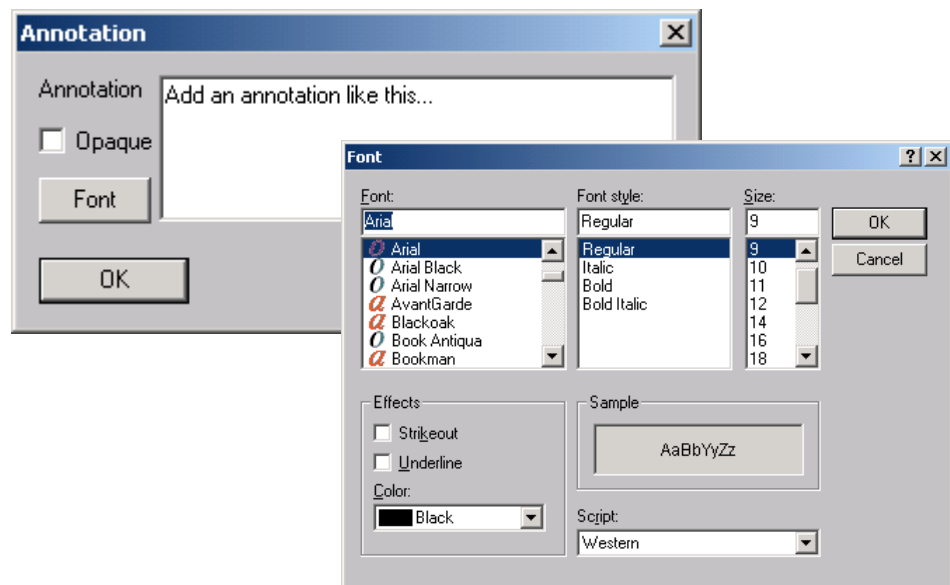
- Black Background — Select this option to display the plot area with a black background.
- Hide Legend — The “legend” is the information at the bottom of the 2D plot that shows the color of the plotted curve and its name. Select this option to hide the legend.
- Show Legend Border — Selecting this option will display a box surrounding the available legend area.

- Auto Tip — Selecting this option causes a popup report of the node infor-



mation to appear when the mouse hovers near a portion of the 2D Curve where there is a node.

- Hide Annotation — Selecting this option hides any annotation added to the plot.
- Add Annotation — Selecting this option causes the Annotation dialog box



to appear. The User can enter a text message, title, note, etc. to be displayed on the 2D Curve plot. The annotation background can be transparent or

opaque. Font attributes of the annotation can be selected by clicking the Font button.

#### NOTE:

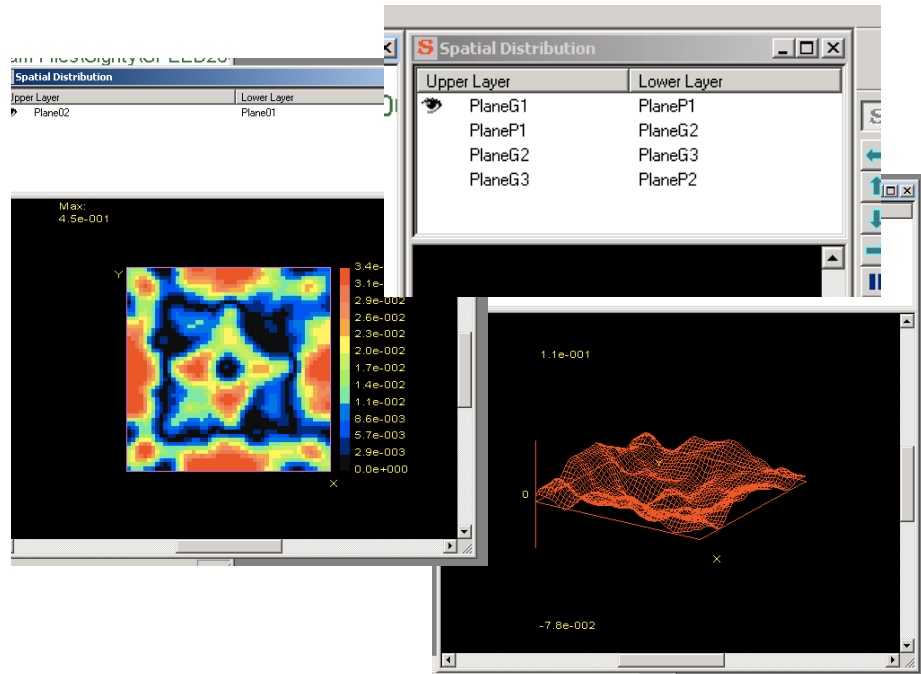
The y-axis label is a special type of annotation. It can be edited as described above. The y-axis label is specified by the user when curves are displayed in the Frequency Domain and/or calculations are performed. The title of the window is an annotation. It can be edited or deleted. By default, it is the path and the name of the current .spd file.

#### Right-click Menus (Frequency Domain)

Right-clicking in the panels of the 2D Curves window when it is in the 2D Curves (Frequency Variation) mode displays the pop up menus with the addition of the FD Calc... option. Refer to [“View Menu \(2D Curves window\)” on page 3-15](#) (FD Calc... menu option) for more information.

#### Spatial Distribution Window

The Spatial Distribution Window permits the user to view the transient simulation of spatial distribution of voltage between planes in real time, in wireframe or color intensity views. The Spatial Distribution window has two panels — the upper panel enables the user to control which layer pair simulations will be displayed in the lower transient simulation data panel. The initial layer display is



determined from the layer stackup information established with SPDGEN. All layer pairs contained in the model will be simulated and all layers with

shapes (or patches) on them will be listed for possible display initially. The user controls which layer pairs will be visible during the transient simulation by clicking on the layer pair list. Click once to enable the display (an open eye appears). Click again to hide the layer pair (the eye closes).

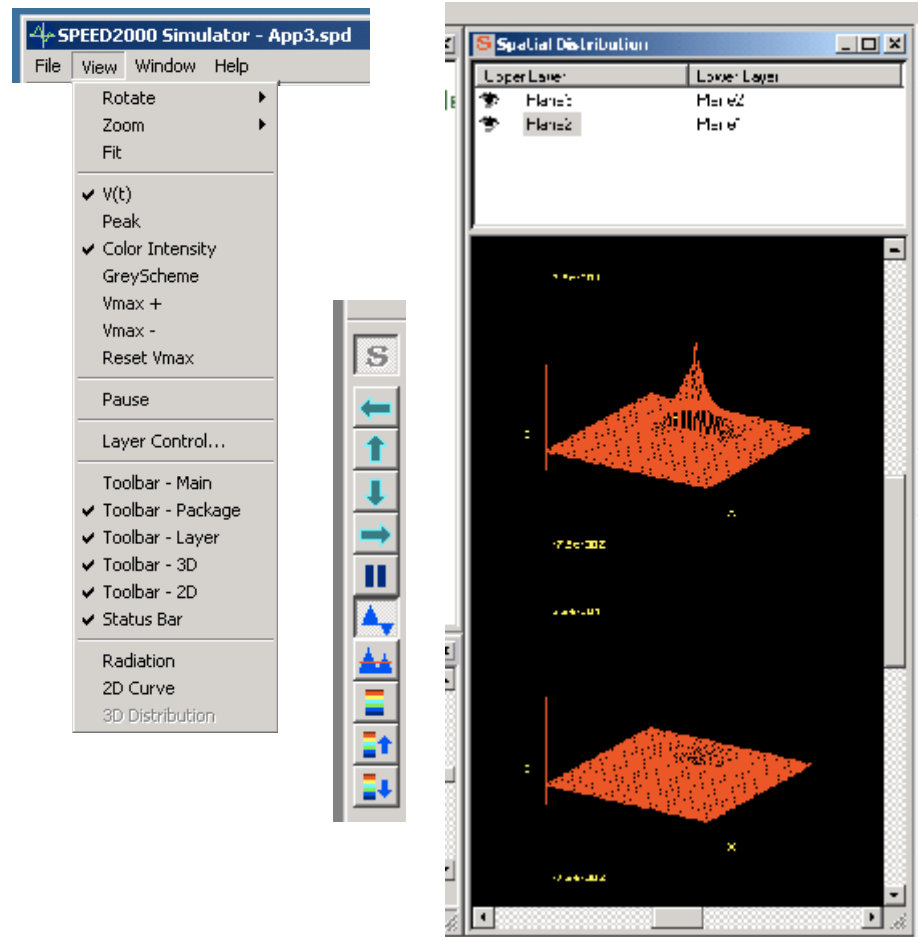
The Spatial Distribution window has a unique menu that is displayed when the window is activated. Options on the File, Window and Help drop-down menus have already been discussed:

- [“File Menu \(after package loaded\)” on page 3-8,](#)
- [“Window Menu” on page 3-12,](#)
- [“Help Menu” on page 3-11.](#)

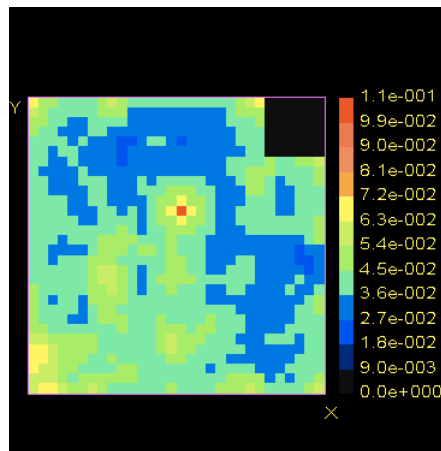
The Spatial Distribution:View Menu is detailed next. Also described in this section is the [“Spatial Distribution Toolbar” on page 3-36.](#)

### View Menu

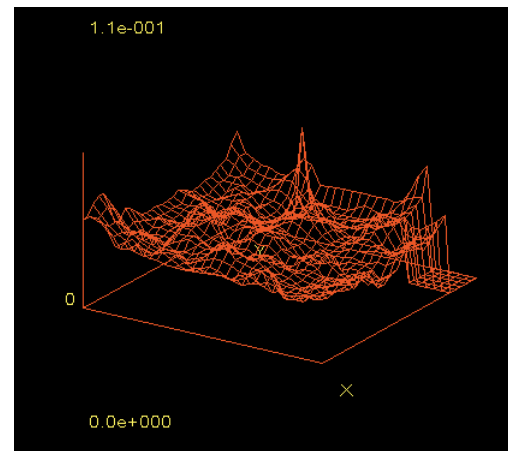
The SPDSIM:View menu has the following options:



- SPDSIM:Spatial Display:View:Rotate — Select this option to rotate the wireframe model about its X and Y axis.
- SPDSIM:Spatial Display:View:Zoom — Select this option to zoom in and out from the wireframe or color intensity model.
- SPDSIM:Spatial Display:View:Fit — Select this option to fit the wireframe or color intensity plot within the viewable window area.
- SPDSIM:Spatial Display:View:V(t) — Select this option to display the voltage values at locations between the planes during the transient simulation. If the Peak/Average voltage distribution between the pair of planes has not been specified in SPDGEN, values of peak/average voltage distribution displayed in the Spatial Distribution window will be zero. Refer to [“Establishing Simulation Parameters” on page 6-1](#).
- SPDSIM:Spatial Display:View:Peak — Select this option to display the maximum voltage values at locations between the planes during the transient simulation.
- SPDSIM:Spatial Display:View:Color Intensity — Select this option to display the color intensity model of the transient simulation. When the Spatial Distribution window is in the Color Intensity mode the simulated voltage at any location can be displayed by right-clicking on the color intensity plot.



color intensity



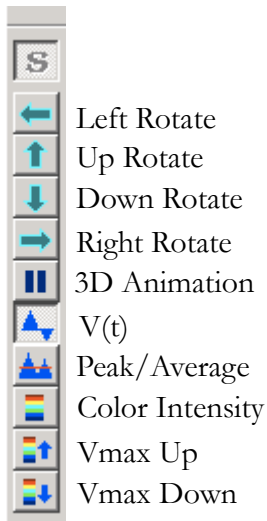
wireframe

- SPDSIM:Spatial Display:View:GreyScheme — Select this option to view the grey scale intensity model of the transient simulation.
- SPDSIM:Spatial Display:View:Vmax+ — Select this option to make the colors of the color intensity model darker relative to your monitor settings.
- SPDSIM:Spatial Display:View:Vmax- — Select this option to make the colors of the color intensity model brighter relative to your monitor settings.
- SPDSIM:Spatial Display:View:Reset Vmax — Select this option to reset the Vmax color range to its default setting.

- SPDSIM:Spatial Display:View:Pause The Pause button in the “S’ tool bar will pause the 3D animation and increase the simulation speed significantly.
- SPDSIM:Spatial Display:View:Layer Control — Not applicable
- SPDSIM:Spatial Display:View:Toolbar -Main — Displays the SPDSIM Main toolbar. Refer to: [“Toolbar - Main \(before a package is loaded\)” on page 3-6.](#)
- SPDSIM:Spatial Display:View:Toolbar - Package — Displays the SPDSIM Package Layer toolbar. Refer to: [“Package Display Toolbar” on page 3-43.](#)
- SPDSIM:Spatial Display:View:Toolbar - Layer — Displays the SPDSIM Layer Display toolbar. Refer to: [“Layer Display Toolbar” on page 3-41.](#)
- SPDSIM:Spatial Display:View:Toolbar - 3D — Displays the SPDSIM Spatial Distribution toolbar. Refer to: [“Spatial Distribution Toolbar” on page 3-36.](#)
- SPDSIM:Spatial Display:View:Toolbar - 2D — Displays the SPDSIM 2D Curve toolbar. Refer to: [“2D Curves Toolbar” on page 3-23.](#)
- SPDSIM:Spatial Display:View:Status Bar. For more information on the Status Bar refer to: [“Status Bar” on page 3-6.](#)
- SPDSIM:Spatial Display:View:Radiation — Displays the Radiation window. For more information, refer to: [“Calculating Radiation” on page 8-1.](#)
- SPDSIM:Spatial Display:View:2D Curve — Selecting this option invokes another 2D Curves window. The additional 2D Curves window permits the user to load a previously generated SPEED curve file (.cur) such that a convenient comparison of previous simulation results are possible.

#### NOTE:

The new 2D Curves window is initially displayed on top of the existing 2D Curves window. Use the pointer to move and reposition the new window as necessary for convenient comparison.



#### Spatial Distribution Toolbar

The SPDSIM:Spatial Distribution toolbar provides convenient access to:

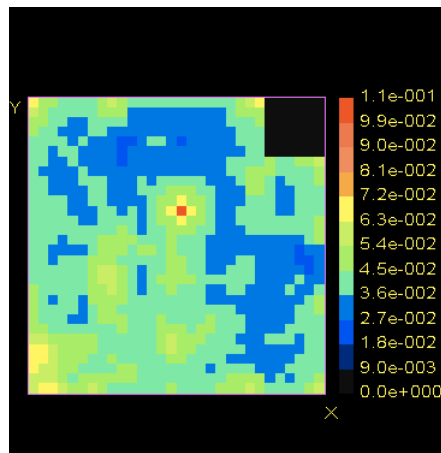
- Left Rotate — click this button to rotate the 3D surface plot left.
- Up Rotate — click this button to rotate the 3D surface plot up.
- Down Rotate — click this button to rotate the 3D surface plot down.
- Right Rotate — click this button to rotate the 3D surface plot right.
- No 3D Animation — click this button to pause the display of the 3D animation. This significantly speeds the simulation time.
- V(t) — click this button to view the spatial transient voltage between layers.

- Peak/Average — click this button to view the spatial Peak or Average Voltages as they are calculated between layers.

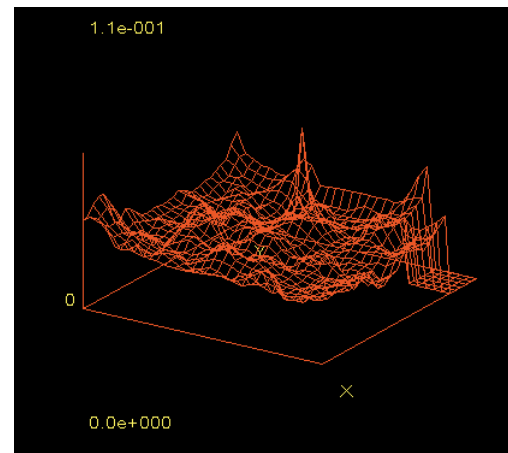
**NOTE:**

The viewing parameter for Peak/Average Voltage is established using SPD-GEN. Refer to [“Choosing Parameters for Voltage Viewing and Analysis” on page 6-18.](#)

- Color Intensity — click this button to display the color intensity plot of spatial voltage distribution.



color intensity



wireframe

- Vmax+ — Click this button to make the colors of the color intensity model darker.
- Vmax- — Click this button to make the colors of the color intensity model brighter.

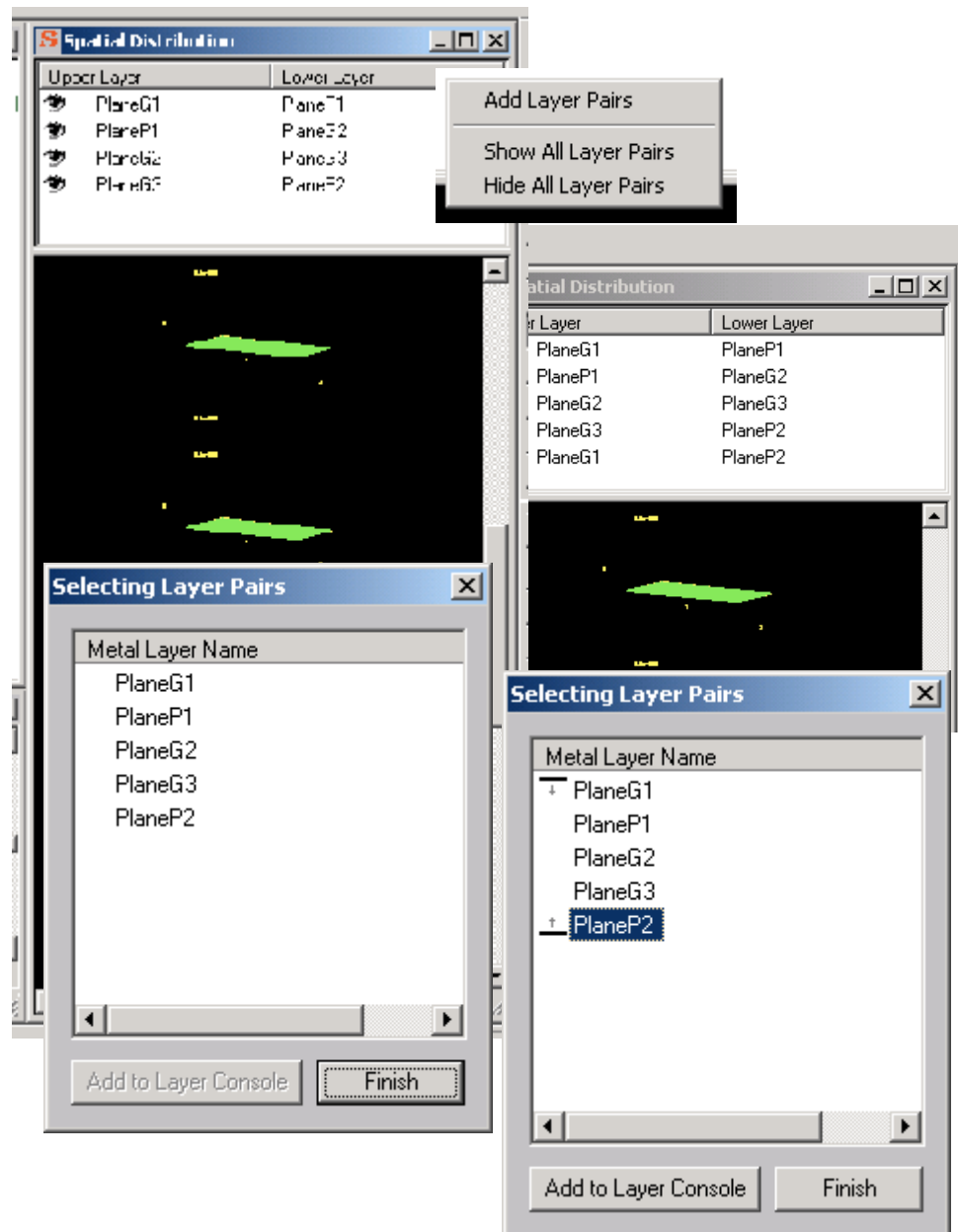
**Right-click Pop-Up Menus (context sensitive)**

Right click in the Spatial Distribution window upper panel to display a context sensitive pop up menu that will enable you to select layer pairs for display.:



- Add Layer Pairs — Select this option to display the Selecting Layer Pairs dialog box. The box contains a listing of the package layers which contain shapes (or patches). All adjacent such layers are initially listed in the Spatial

Distribution window upper panel — this interface permits the user to select layers for simulation in addition to the adjacent layers. Click on a layer to select the upper member of the pair, then click on another layer to select the lower member of the pair. Click the Add to Layer Console button to add the layer pair to the Spatial Distribution upper panel listing. Click the Finish button when you are done.



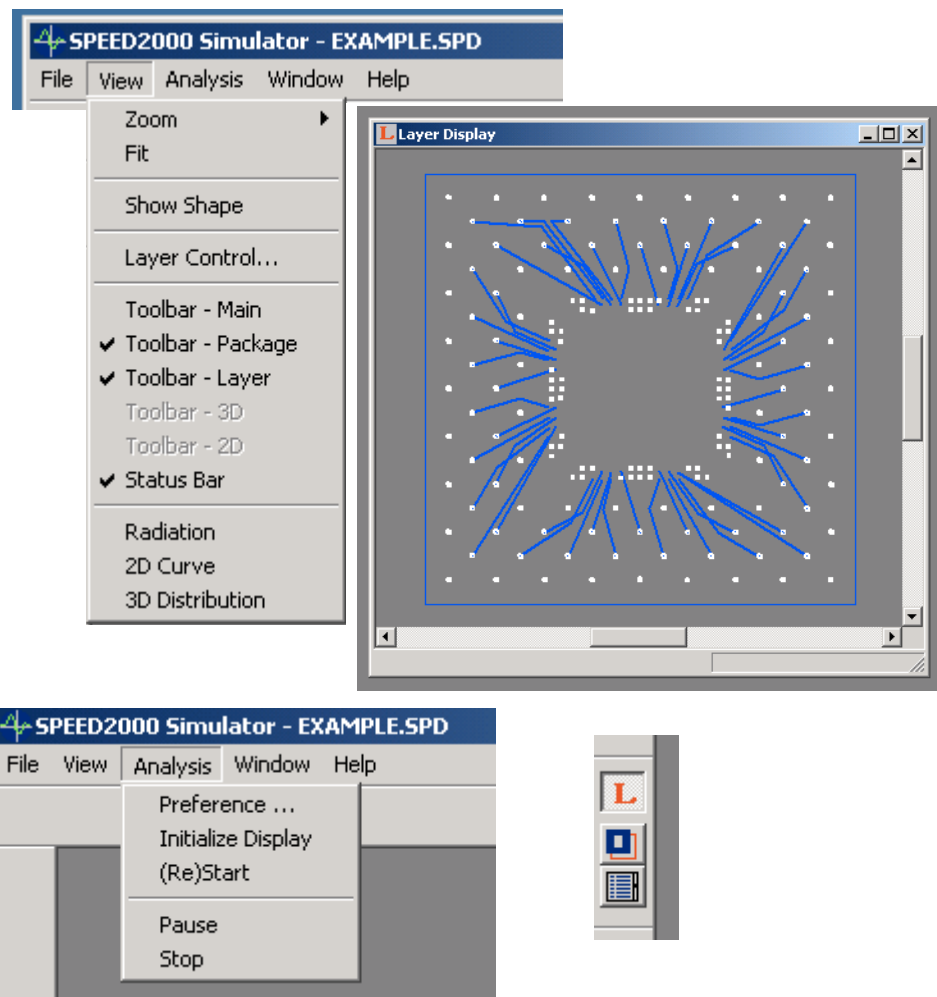


**NOTE:**

The calculation and simulation of non-adjacent layer pair data is derived from adjacent pair transient simulation calculations.

**Layer Display Window**

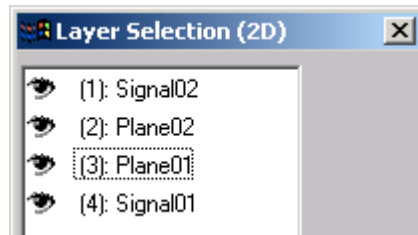
The SPDSIM:Layer Display window provides a convenient top down view of the package components selectable on a per layer basis. The SPDSIM:Layer Display window has a unique toolbar and two drop down menus — View and Analysis. The Layer Display window's View and Analysis menus and toolbar are discussed in this section. Please refer to previous discussions for details on: [“File Menu \(after package loaded\)” on page 3-8](#), [“Window Menu” on page 3-12](#) and [“Help Menu” on page 3-6](#).



**View Menu**

The SPDSIM:Layer Display:View menu has the following options:

- SPDSIM:Layer Display:View:Zoom — Select this option to zoom in/out from the Layer view.
- SPDSIM:Layer Display:View:Fit — Select this option to fit the package model in the Layer Display window.
- SPDSIM:Layer Display:View:Show Shape — Select this option to display package shapes and traces. Use this option in conjunction with the Layer Control option to selectively display package components on a layer.
- SPDSIM:Layer Display:View:Layer Control — Select this option to display the layer selection window that enables the user to selectively display the package layers. Click on an open eye to hide a package layer. Click on a closed eye to show a layer. Use this option in conjunction with the Show Shape option to selectively display package shape components on a layer.



- SPDSIM:Layer Display:View:Toolbar -Main — Displays the SPDSIM Main toolbar. Refer to: [“Toolbar - Main \(before a package is loaded\)” on page 3-6](#).
- SPDSIM:Layer Display:View:Toolbar - Package — Displays the SPDSIM Package Layer toolbar. Refer to: [“Package Display Toolbar” on page 3-43](#).
- SPDSIM:Layer Display:View:Toolbar - Layer — Displays the SPDSIM Layer Display toolbar. Refer to: [“Layer Display Toolbar” on page 3-41](#).
- SPDSIM:Layer Display:View:Toolbar - 3D — Displays the SPDSIM Spatial Distribution toolbar. Refer to: [“Spatial Distribution Toolbar” on page 3-36](#).
- SPDSIM:Layer Display:View:Toolbar - 2D — Displays the SPDSIM 2D Curve toolbar. Refer to: [“2D Curves Toolbar” on page 3-23](#).
- SPDSIM:Layer Display:View:Status Bar. For more information on the Status Bar refer to: [“Status Bar” on page 3-6](#).
- SPDSIM:Layer Display:View:Radiation — For more information, refer to [“Calculating Radiation” on page 8-1](#).
- SPDSIM:Layer Display:View:2D Curve — Selecting this option invokes another 2D Curves window. The additional 2D Curves window permits the

user to load a previously generated SPEED curve file (.cur) such that a convenient comparison of previous simulation results are possible.

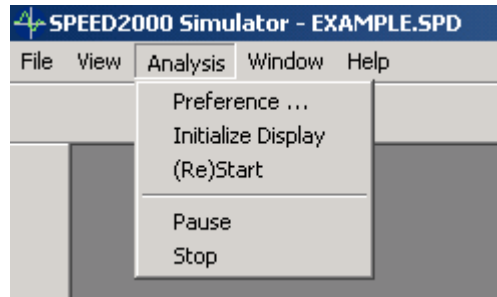
#### NOTE:

The new 2D Curves window is initially displayed on top of the existing 2D Curves window. Use the pointer to move and reposition the new window as necessary for convenient comparison.

- SPDSIM:Layer Display:View:3D Distribution. When the Spatial Distribution window is closed, this option is active. Selecting it will open the Spatial Distribution window. For more information refer to: [“Spatial Distribution Window” on page 3-33.](#)

#### Analysis Menu

The SPDSIM:Layer Display:Analysis menu has the following options:



- SPDSIM:Layer Display:Analysis:Preference — Select this option to display the Preference Dialog Box. For detailed information see [“Tools Menu” on page 3-17.](#)
- SPDSIM:Layer Display:Analysis:Initialize Display — This option is not activated with this release of SPDSIM.
- SPDSIM:Layer Display:Analysis:(Re)Start — Select this option to re-start the transient simulation.
- SPDSIM:Layer Display:Analysis:Pause — Select this option to pause the transient simulation.
- SPDSIM:Layer Display:Analysis:Stop — Select this option to stop the transient simulation.



#### Shape On/Off Layer Control

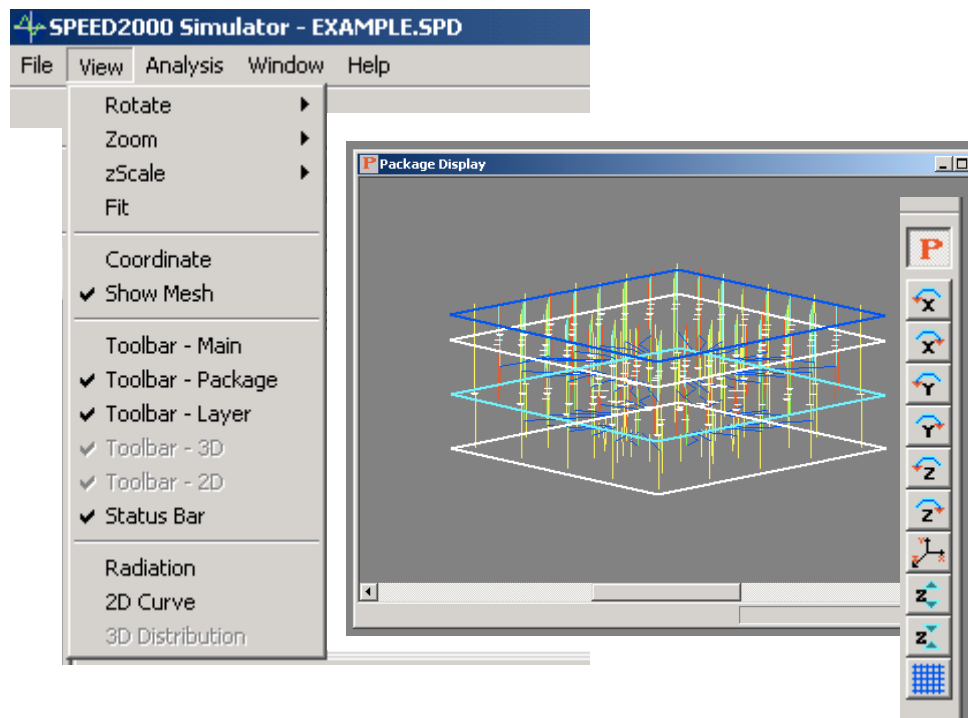
The SPDSIM:Layer Display toolbar provides convenient access to:

- Shape On/Off — Click this button to selectively package shapes.
- Layer Control — Click this button to display the layer selection window that enables the user to selectively display the package layers. Click on an open eye to hide a package layer. Click on a closed eye to show a layer. Use

this option in conjunction with the Show Shape option to selectively display package components on a layer.

### **Package Display Window**

The SPDSIM:Package Display window provides a convenient three dimensional view of the package — rotatable on the X, Y, Z axis. The SPDSIM:Package Display window has a unique toolbar and two drop down menus — View and Analysis. The Package Display window's View menu and toolbar are discussed in this section. Please refer to previous discussions for details on: [“File Menu \(after package loaded\)” on page 3-8](#), [“Analysis Menu” on page 3-41](#), [“Window Menu” on page 3-12](#) and [“Help Menu” on page 3-6](#).



### **View Menu**

The SPDSIM:Package Display:View menu has the following options:

- SPDSIM:Package Display:View:Rotate — Select this option to display a fly-out menu with X, Y and Z axis rotation options.
  - X Counter Clockwise
  - X Clockwise
  - Y Counter Clockwise
  - Y Clockwise
  - Z Counter Clockwise



X Counter Clockwise  
X Clockwise  
Y Counter Clockwise  
Y Clockwise  
Z Counter Clockwise  
Z Clockwise  
Display Coordinate  
Z Increase  
Z Decrease  
Mesh On/Off

- Z Clockwise

- SPDSIM:Package Display:View:Zoom — Select this option to zoom in/out from the Package view.
- SPDSIM:Package Display:View:zScale — Select this option to display the flyout menu to increase and decrease the package layers z-scale in a z direction..
- SPDSIM:Package Display:View:Fit — Select this option to fit the package model in the Package Display window.
- SPDSIM:Package Display:View:Coordinates — Select this option to display the X, Y and Z axis icon relative to the coordinate system.
- SPDSIM:Package Display:View:Show Mesh — Select this option to display the numerical mesh used in computation.

### Package Display Toolbar

The SPDSIM:Package Display toolbar provides convenient access to:

- SPDSIM:Package Display:toolbar:X Counter Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:X Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Y Counter Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Y Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Z Counter Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Z Clockwise — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Display Coordinate — Click this button to rotate the package about the X-axis.
- SPDSIM:Package Display:toolbar:Z Increase — Click this button to increase the z-axis distance between the layers.
- SPDSIM:Package Display:toolbar:Z Decrease — Click this button to decrease the z-axis distance between the layers.
- SPDSIM:Package Display:toolbar:Mesh On/Off — Click this button to display the package mesh.



## Working with the Package

Typically, a three dimensional packaging structure will be brought into SPEED2000 through a translator, if it is built by a physical layout tool. However, SPEED2000 also provides powerful 3D editing functions so that you can easily:

- modify an existing layout file;
- or create a structure manually.

In the first scenario, after translating a layout file into Sigrity native file format (.spd), you may want to verify the translated structure using [“Finding Objects” on page 4-113](#), checking via properties (see [“Working on Vias” on page 4-68](#)), checking trace properties (see [“Working on Traces” on page 4-91](#)) and other functions. Then you may wish to modify the structure by adding or deleting some of the objects, simplifying the plane shapes, etc.

In the second scenario or to build a structure from scratch, you will need to get familiar with the drawing/editing functions provided by the software such as:

- Setting up the layer stack-up for package/board layers;
- Creating shapes for metal planes and patches;
- Adding vias;
- Adding traces;
- Checking for drawing errors.

The following sections explain procedures that you need to know to work with packages.

## Opening, Closing and Saving Files



Open Button

### ► Opening an existing .spd file:

1. Click File:Open from the menu bar or click the *Open* button on the tool bar of the software.
2. In the Open dialog box that appears, browse to select the file to be loaded.

3. Click the *Open* button to exit the Open dialog box. Your file will be opened.

**NOTE:**

You can work with only one .spd file at a time.

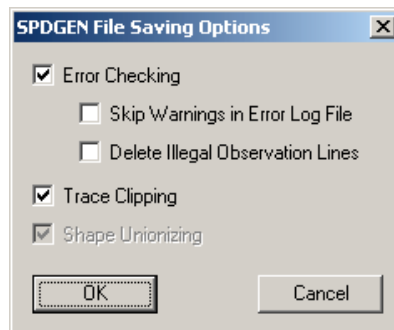
**► Closing/Exiting a .spd file:**

- If you just want to close the current project without quitting the software, click File:Close from the menu bar. You will be prompted to save the current project — if changes to the current project have not been saved.
- If you want to close the file and exit the software, click File:Exit from the menu bar. If changes to the current project have not been saved you will be prompted to save the current project before exiting the software.

**File Saving Options Dialog Box****NOTE:**

During the file saving process, you may be prompted by the SPEED2000 File Saving Options dialog box.

The File Saving Options dialog box appears when the user saves or exits a file. It permits the user to select several .spd file saving options:



- File Saving Options:Error Checking option — enable error checking during this file save. The error log file is saved as *filename\_SPEED2000.err*. For more information on Error Checking, please refer to [“Checking Drawing Errors” on page 4-99](#).
- File Saving Options:Skip Warnings in Error Log File — only output the error messages, not the warning messages.



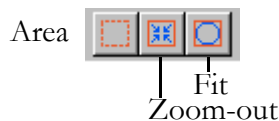
- File Saving Options:Delete Illegal Observation Lines — delete invalid viewing parameters, which may exist after the objects no longer exist in the .spd file.
- File Saving Options:Trace Clipping — when a trace overlaps a patch, the overlapping portion will be clipped. For more information, refer to [“Pre-Conditioning the .spd File For Simulation” on page 4-55](#), [“To clip traces and unionize shapes:” on page 4-59](#) and [“To segment and clip traces, unionize shapes, highlight special voids:” on page 4-61](#).
- File Saving Options:Shape Unionizing — combine overlapping polygons into one unique polygon. For more information, refer to [“Pre-Conditioning the .spd File For Simulation” on page 4-55](#), [“To unionize shapes \(only\):” on page 4-55](#), [“To clip traces and unionize shapes:” on page 4-59](#) and [“To segment and clip traces, unionize shapes, highlight special voids:” on page 4-61](#).

### Auto Save

The .spd file will be automatically backed up every 10 minutes. The time interval can be changed in Tools:Options:Auto Save. The name of the back up file will be your .spd file + .auto extension within the same directory as your .spd file. This “.auto” file is replaced by .spd file, when a normal file save operation is completed.

## Using Zoom Functions

Several ZOOM functions are provided in the software.

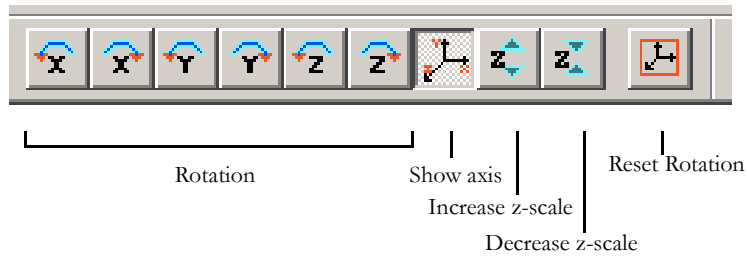


- Area: display a magnified view of a rectangular area selected by mouse dragging. This enables you to conveniently Zoom-in on a selected area.
- Zoom-out: display the window contents in reduced size.
- Fit: display the entire curve graph or package in the window.

## Viewing the Structure in 3D

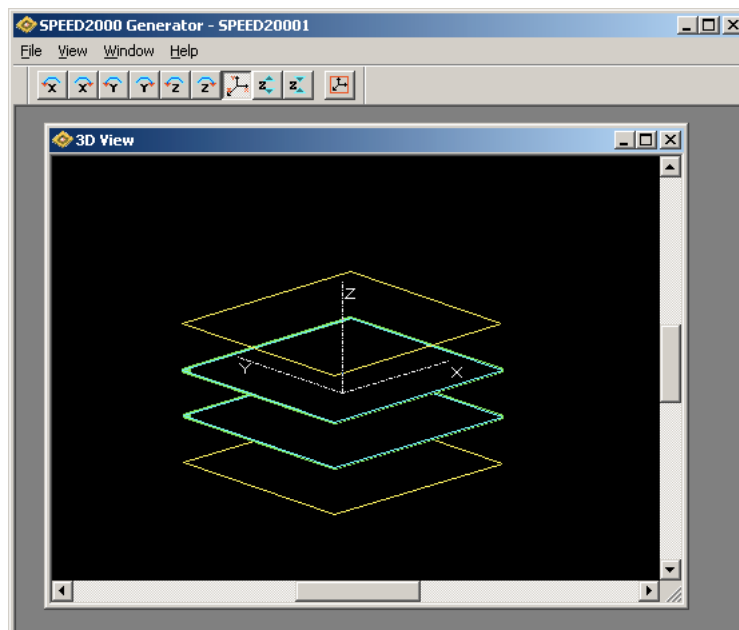
Click View:3D View from the menu bar of the software. A 3D View window is opened with the 3D view of the package structure.

Click View:Toolbar-3D from the menu to unfold the 3D Viewing Tools buttons as shown below.



You can use the rotation buttons, scale buttons, and zoom buttons to reveal the details of the 3D structure. The four arrow keys on the keyboard can also be used for the rotation of the 3D structure.

Click View:Show Shape from the menu bar to switch between the filled-pattern view and the outline view.



### NOTE:

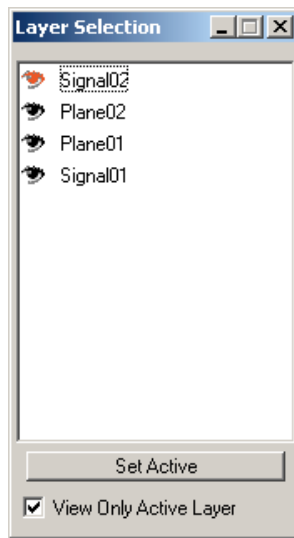
Depending on whether the 3D View window or the Layer View window is active, items on the menu bar will be different.

## Viewing Layers and the Layer Selection Dialog Box


The Layer View window is the place where you can see the top-down view of the package and add or modify components on a selected layer. The Layer Selection dialog box is used to control which layer(s) — and their contents — can be seen and which layer you want to work on in the Layer View window.

### ► To open the Layer Selection dialog box:

1. Click View:Layer Selection from the menu bar, the Layer Selection dialog box appears.



### NOTE:

The dialog box can be closed by clicking the button at the upper-right corner of the dialog box. If you click the *Minimize* button  you can roll up the Layer Selection dialog box so that it will not block your view.

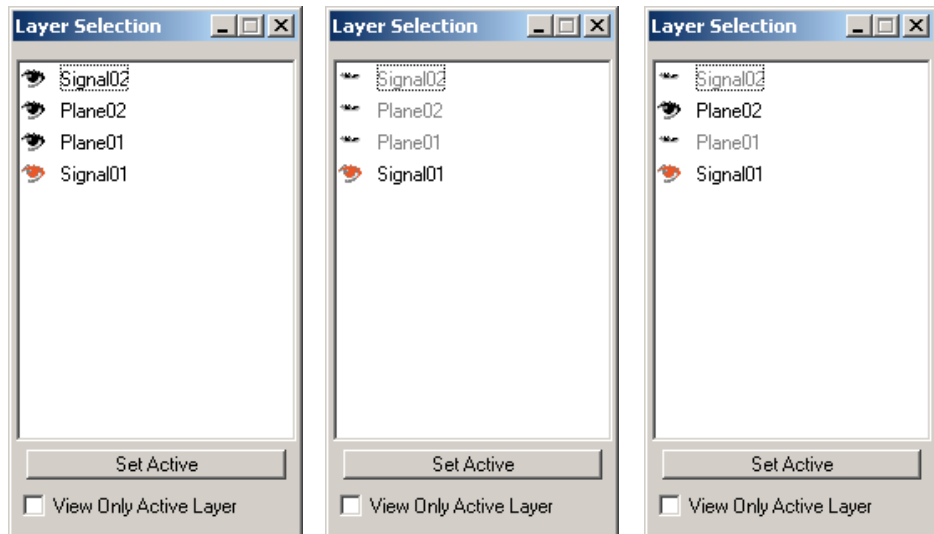
### Tip:

Two hot keys are set to help you use the zoom functions. F2 is for area zoom and F3 is for zoom out.

### ► To select layers for viewing:

1. Click View:Layer Selection from the menu bar, the Layer Selection dialog box appears.
2. Click the box in front of the **View Only Active Layer** option, to add a checkmark, if you just want to view the components of the active layer. Click the box in front of the View Only Active Layer option, to remove the checkmark, if you want to view the components of all layers.
3. Click on the names of layers in the Layer Selection dialog box to switch the individual viewing mode of layers between On and Off. The viewing mode of a layer is “On” if its name is in bold and has an open-eye icon to the left of its name.

The viewing mode is “Off” if its name is in gray and it has a close-eye icon in front of it. When the viewing mode of a layer is on AND the **View Only Active Layer** option is off, components on this layer are displayed in the Layer View window.



The contents of all layers can be viewed on the Layer View window.

The contents of layer Signal01 can be viewed on the Layer View window.

The contents of layers Signal01 and Plane02 can be viewed.

#### NOTE:

The “active layer” cannot be placed into the “Off” viewing mode.

#### Display of Shape Modes

The display of shapes on plane layers in the Layer View window has two modes — *the outline view* and *the filled pattern view*. You can switch between the two modes by clicking View>Show Shape from the menu bar. To see layers in an outline view, make sure the View>Show Shape item from the menu bar is not checked. The outline view shows a rectangular outline of the layer. If the View>Show Shape menu has a check mark, you will see the actual shape of the metal plane with metal areas filled with a certain color. In this mode, the view of the objects on layers underneath the metal plane is blocked.

► **To activate a layer for drawing/editing:**

If you want to add or edit objects on a layer, that layer has to be activated first.

**NOTE:**

Only one layer can be set active at a time.

1. Click View:Layer Selection from the menu bar, the Layer Selection dialog box appears.
2. Click the *Set Active* button, then click the name of the layer that you want to activate in the Layer Selection dialog box. You will notice that the active layer will have a red-eye icon in front of it.

## Setting Up the (Layer) Stack-up

The Stack-up dialog box shows the layer information of a package. In this dialog box, you can add or delete three types of layers:

- Signal layer - on which metal signal traces or patches are placed
- Plane layer - solid metal planes of various shapes
- Dielectric layer - insulating material separating metal layers

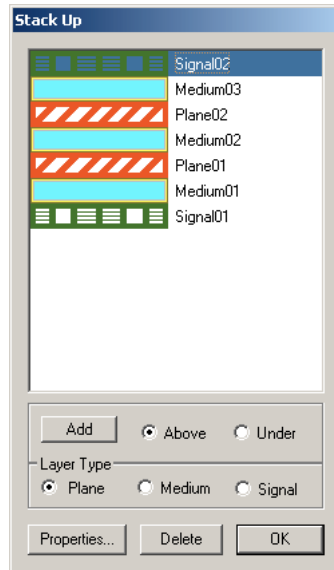
Within our software, plane layers can only contain shapes, not traces, whereas the signal layer can contain both traces and shapes. Shapes on a signal layer are referred to as “patches”.

► **To open the stack-up dialog box:**

1. Click Edit:Stack Up from the menu bar of the software, the *Stack Up* dialog box appears. In the *Stack Up* dialog box, all the layers of a package are displayed in the top-to-bottom order.

**NOTE:**

For a new file, the default setup is a 4-layer structure with one signal layer on the top, two solid metal planes in the middle, and one signal layer at the bottom.



► **To add a layer:**

1. In the *Stack Up* dialog box, highlight the layer (by clicking on the layer) to which the layer to be added is adjacent.
2. Select the type (Plane, Medium or Signal) of the layer to be added in the Layer Type field.
3. Select to add either Above or Under the highlighted layer.
4. Click the *Add* button.

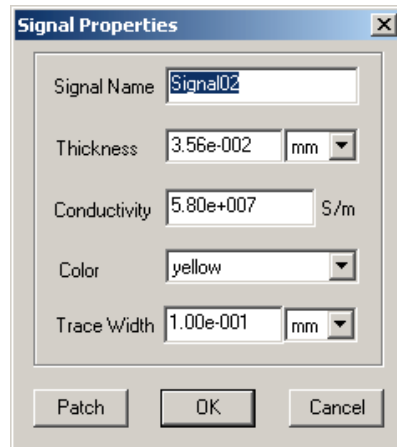
**NOTE:**

Two metal layers (either signal or plane) can not be adjacent to each other. Make sure a dielectric layer placed between two metal layers is added first.

5. Set properties of the new layer by following instructions in the sections below.

► **To set properties of a signal layer:**

1. Double click on the signal layer in the Stack Up dialog box; or, highlight the layer, then click the *Properties* button in the Stack Up dialog box. The Signal Properties dialog box appears.



2. Modify the signal layer name if needed. The name has to start with “Signal”
3. Change the default metal thickness of the signal layer as needed.
4. Change the conductivity of the metal as needed.
5. Set the default color for all the traces on that signal layer. .

**NOTE:**

If there are more than one signal layers, you may want to set different colors to distinguish different signal layers.

To set the color of an individual trace, see the instructions in the section [“Working on Traces” on page 4-91](#).

6. Change the default width of signal traces as needed.

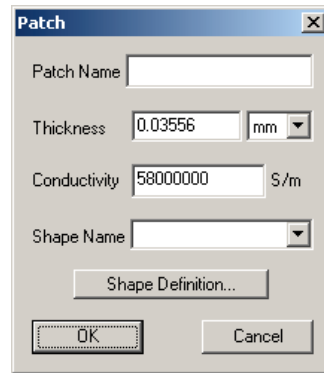
**NOTE:**

When traces are placed on the layer, they will automatically have the default trace width, and the default trace thickness which is the same as the thickness of the corresponding signal layer.

To set the width of an individual trace, see the instructions in the section [“Working on Traces” on page 4-91](#).

7. If you click on the *Patch* button, a Patch dialog box will appear. A shape on a signal layer is called a “patch”. Working with patches is similar to operations on

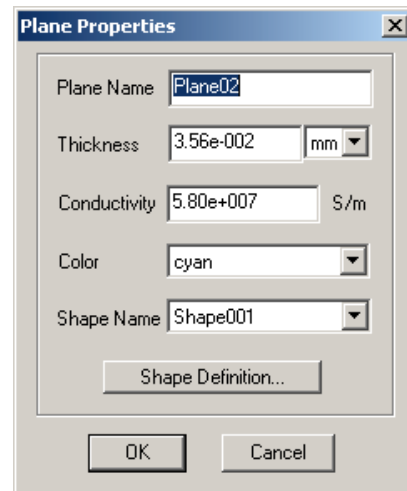
shapes on plane layers. Refer to [“To set properties of a plane layer:” on page 4-10](#), [“Setting Up the \(Layer\) Stack-up” on page 4-7](#) and [“Working on Shapes” on page 4-29](#).



8. Click OK to exist the Patch dialog box.
9. Click the *OK* button to exit the Signal Properties dialog box.

► **To set properties of a plane layer:**

1. Double click the plane layer in the Stack Up dialog box; or, highlight the layer, then click the *Properties* button in the Stack Up dialog box. The Plane Properties dialog box appears.



2. Modify the plane layer name if needed. The name must start with “Plane”.
3. Change the thickness of the metal plane as needed.
4. Change the conductivity of the metal as needed.
5. Select a color for the outline of this metal plane.



6. Select a shape name for the plane.

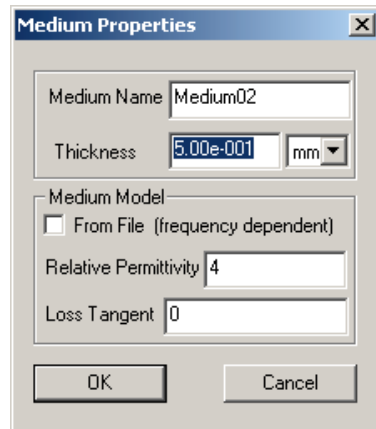
**NOTE:**


A shape name corresponds to a pre-defined shape definition for metal planes. More detailed descriptions on how to work with shapes can be found in the section [“Working on Shapes” on page 4-29](#).

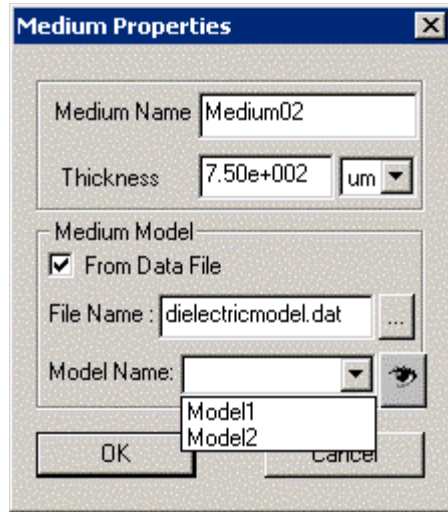
7. Click the *OK* button to exit the Plane Properties dialog box.

► **To set properties of a dielectric (medium) layer:**

1. Double click the dielectric layer in the Stack Up dialog box; or, highlight the layer, then click the *Properties* button in the Stack Up dialog box. The Medium Properties dialog box appears.



2. Modify the medium name if needed. The name must start with “Medium”.
3. Change the thickness of the dielectric medium as needed.
4. A constant relative permittivity and loss tangent for the dielectric material can be entered in the dialog box. Or, frequency-dependent relative permittivity and loss tangent can be entered through a data file. Check the box preceding *From File*, the Medium Properties dialog box is changed as shown in the figure below. Enter the name of the file containing the model (or click ... button to browse); then select the model from the Model Name listing. The data and curves of the relative permittivity and the loss tangent in the selected model can be viewed by clicking the  button.



**NOTE:**

A data file may contain more than one dielectric models. All the dielectric models in a selected file are listed in the box of Model Name.

The format of the dielectric model file is shown below:

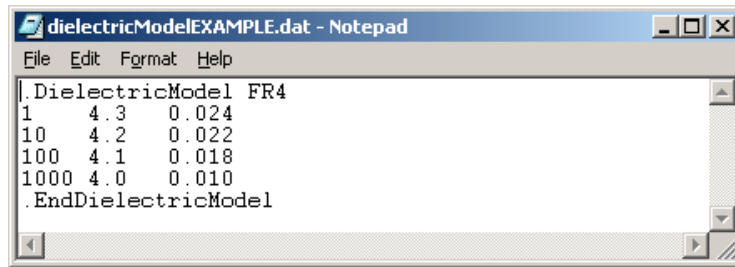
```
. DielectricModel Modelname
  f1(MHz)      εr1      tan δ1
  f2(MHz)      εr2      tan δ2
  .
  .
  fn(MHz)      εm      tan δn
.EndDielectricModel
```

**NOTE:**

The dielectric model below  $f_1$  and beyond  $f_n$  are assumed as follows:

For  $f < f_1$ ,  $\epsilon_r = \epsilon_{r1}$      $\tan \delta = \tan \delta_1$   
 For  $f > f_n$ ,  $\epsilon_r = \epsilon_{rn}$      $\tan \delta = \tan \delta_n$

5. Click the *OK* button to exit the Medium Properties dialog box. Following is an example of a dielectric model file.



► **To delete a layer:**

1. Highlight the layer, which can be either a signal, plane or dielectric (medium) layer, in the Stack Up dialog box.
2. Click the *Delete* button in the Stack Up dialog box.

**NOTE:**

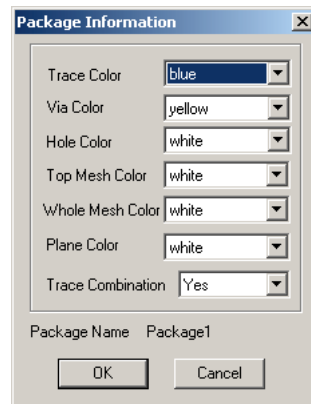
**A plane layer can only be deleted if there are no vias connected to the plane.** Therefore, if you really want to delete the plane layer, delete all the vias connected to the plane first.

**A signal layer can only be deleted if there are no traces on the layer and there are no vias terminated at the layer.** If you really want to delete the signal layer, delete all the traces on the layer and all the vias connected to the layer first.

A dielectric layer can only be deleted if its removal will not cause two metal layers in direct contact with each other. A metal layer, which can be a signal or plane layer, may need to be deleted first before deleting a dielectric layer.

► **To establish/change the drawing colors:**

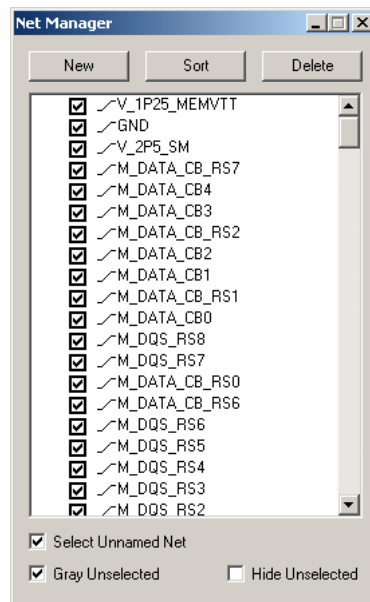
1. Click Tools:Options:Component Colors... from the menu bar of the software, the Package Information dialog box appears.



2. Select the color of the various objects from the drop down menus.

## Net Management

The elements of a package are often organized on the basis of “nets” — where each package object (node, shape, via, trace) is associated with a “net name”. The “net” is generally understood to consist of all those package elements associated with a signal and its physical path through the package. The Net Manager provides an interface to enable the User to conveniently manage package elements on the basis of the net they are associated with. The Net Manager is also used to select and deselect nets to be simulated.



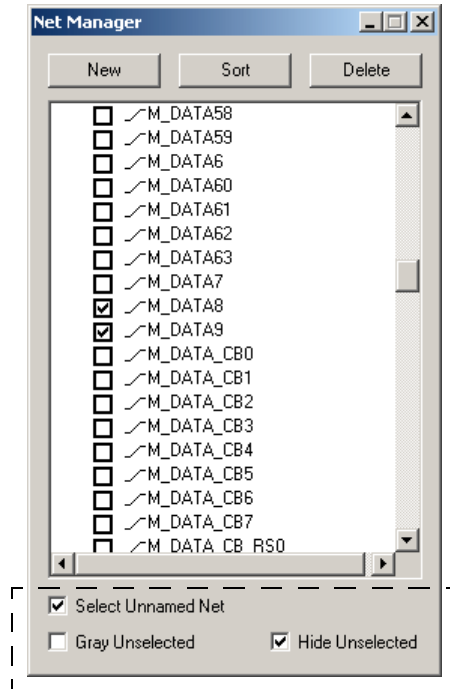
With the Layer View window activated, select View:Net Manager. The Net Manager dialog box will be displayed. Shown in the working area are all of the Net Names present within the .spd file.

### “Unnamed” Nets

Package objects without a declared Net Name present in the .spd file are internally associated with a default Net Name of “Unnamed” net. Thus, all package objects can be conveniently managed through the Net Manager.

► **To view and simulate package components on the basis of Nets:**

1. With the Layer View window activated, select View:Net Manager. The Net Manager dialog box will be displayed.



Three checkboxes, at the bottom of the Net Manager, plus the checkboxes preceding the net names listed, give the user complete control over the viewing and simulation of nets.

Select Unnamed Net: highlights objects without net names.

Gray Unselected: when checked, dims nets not selected for viewing or simulating.

Hide Unselected: when checked, hides nets not selected for viewing or simulating.

► **To deselect nets selected:**

Hold down the control key to deselect (remove the checkmark) from individual nets. Hold down the shift key to select/deselect groups of nets for viewing.

► **To select all objects associated with the unnamed net:**

The “Select Unnamed Net” checkbox highlights all objects associated with the “Unnamed” Net. See the figure above.

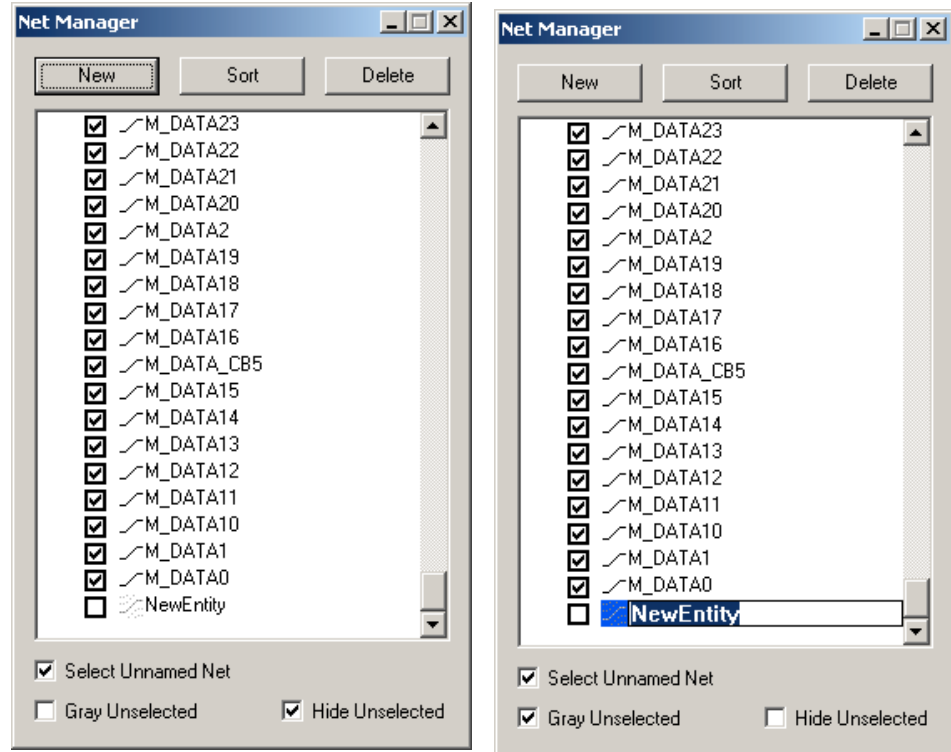
► **To mute (or gray out) objects unselected:**

Nets not selected for viewing can be “grayed” out or hidden entirely, using the “Gray Unselected” checkbox or the “Hide Unselected” checkbox. See the figure above.

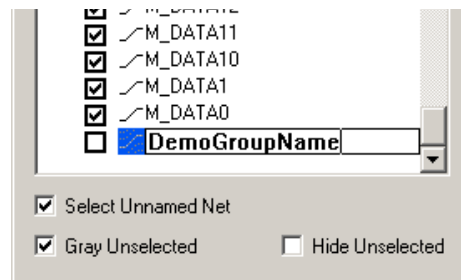
► **To organize nets within a hierarchy:**

Nets can be organized into hierarchical groups for convenient management.

1. Press the *New* button. Observe the entry made to the bottom of the net listing, “NewEntity”. The muted lines preceding the new entry indicate this new entity is available to become a group name entity.

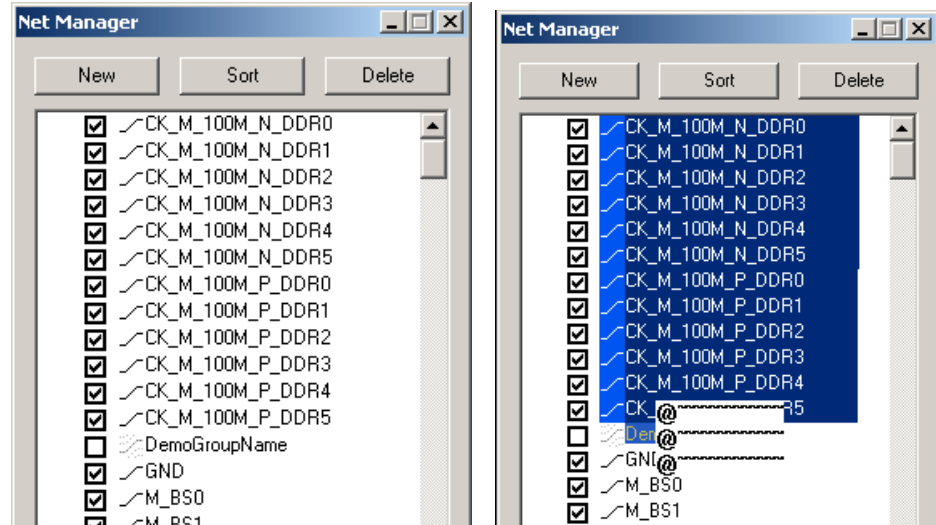



2. Click on the name of the NewEntity to activate the name field.
3. Enter a new name for the entity using the keyboard. Our example uses *DemoGroupName*.

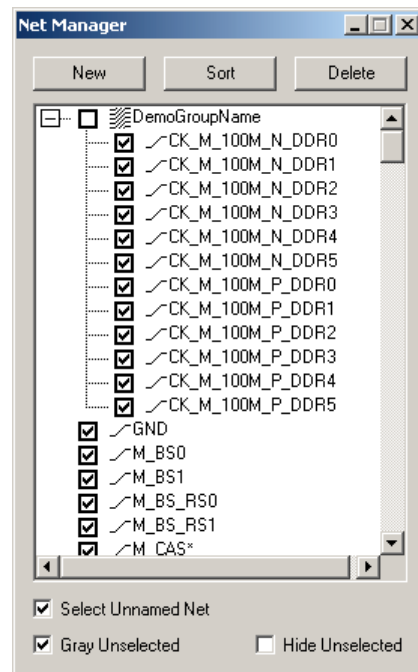


4. Add members to the new group identity, *DemoGroupName*. Press the *Sort* button to alphabetically sort the Net Name list. Select members for the new group one-at-a-time or several (hold down the shift key) and drag-and-drop the selected



members to the new group name, *DemoGroupName*. Observe the new group name becomes highlighted during the drag-and-drop operation to indicate it is activated to accept the drop.

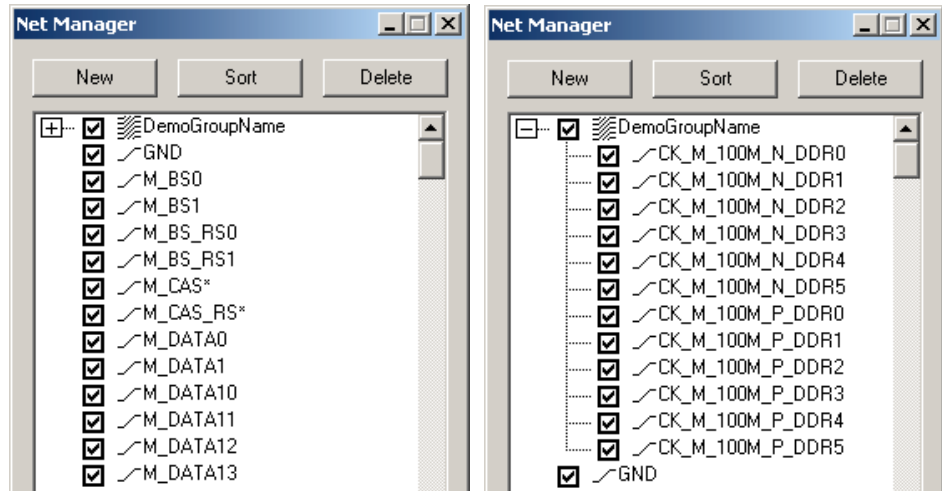


5. Observe the net names “dragged” to the group name are now placed under the group name, the icon preceding *DemoGroupName* is altered to a more vivid hue, designating this entity as a group name, and the Windows icon, , indicating Windows “hierarchy tree logic” now precedes the *DemoGroupName*.





6. Click the tree symbol, , to contract the group. Click the tree symbol, , to expand the group.



undo button

#### NOTE:

When necessary, use the *undo* key (available on the main toolbar) to undo any previous actions.

#### ► To sort nets:

Click the *Sort* button to alphabetically sort the listed nets in ascending order — click again to sort in descending order.

#### ► To delete nets:

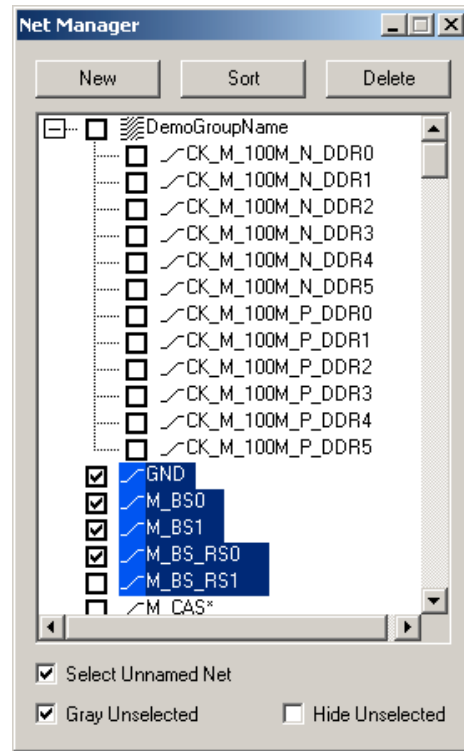
Objects can be deleted from the .spd file on the basis of their net association.

#### NOTE:

Observe in the figure, the name field is highlighted in blue in conjunction with the delete function. The checkboxes are not used in conjunction with the delete function.

1. Select a net name or a series of net names from the net name list. Use the shift and control keys in conjunction with pointing and clicking to select the nets of interest. For example, hold down the control key and click on net names to individually select the net names. Hold down the shift key, after selecting the initial starting point in the list, then select the ending name of contiguous net names of interest, to select several nets at once.

2. Click on the *Delete* button. The selected nets will disappear from the net list and be deleted.

**NOTE:**

All objects associated with a deleted net will be deleted from the .spd file.

► **Adding new net names to objects:**

Objects not explicitly associated with a net name are internally associated with the “unnamed” net group. Package objects can be explicitly associated with a net name by editing their Net Name object property. Assigning a new net name to an object in this manner, adds the new net name to the net name listing (seen in the Net Manager workspace) and makes the object available for manipulation within the Net Manager. Refer to [“Working on Shapes” on page 4-29](#) for more information about modifying object properties.

## Showing Objects

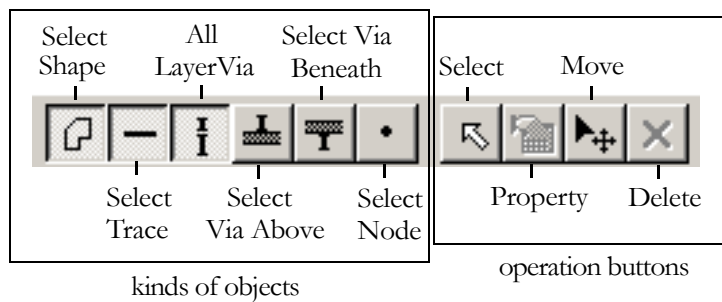
Several Layer View menu options enable the User to conveniently highlight objects of a package. As well as providing the big picture, “showing objects” facilitates further zooming down for closer inspection on a small area.

- View:Show Nodes — Highlights the package nodes.

- View:Show Shape — Highlights the package shapes.
- View:Show Grid — Highlights the package grid.
- View:Show All Couple Lines — Highlights all Couple Line objects.
- View:Show Connected Circuit — Highlights all connected circuits.
- View:Show Special Voids — Highlights all Special Voids.
- View:Show Via Node Contact — Highlights all contacts of via and node.

## Selecting Objects

Sometimes, the object you want to select for a procedure is so close to another object on a layer that it is challenging to select “just that object” — or a certain kind of object. A special toolbar exists in SPDGEN to help you select specific objects:



### ► To select certain kinds of objects:

The general procedure is to...

1. Click on the button or buttons of the kinds of objects that you want to select; note in the figure above there is a button for shapes, traces, vias (on all layers, above the active layer and below the active layer) and nodes.
2. Then click the operation button:
  - *Select* — move the cursor to the objects of interest, click the left mouse button, hold and drag to select an object or objects.
  - *Property* — move the cursor to the object of interest and click the left mouse button to select an object. Then click the *property* button to see the property report for the object.
  - *Move* — move the cursor to the objects of interest and click the left mouse button and hold. The select object/objects will move with the Move cursor.

- *Delete* — move the cursor to the objects of interest, click the left mouse button, hold and drag to select an object or objects. Click the delete key to remove the object/objects.

**NOTE:**

To DE-select an object, hold down the *Ctrl* key and click on the object selected.

## Setting Up the Display, Drawing and Mesh Environment

The x-y coordinates of the cursor are shown at the lower-right corner of the main program window. The unit of the displayed X and Y coordinates can be set to your preferences. The grid can give you visual help during drawing, and the Snap To Grid/Mesh option can further help you to draw objects at precise locations. With the Snapping To option you can locate objects directly onto mesh or grid points.

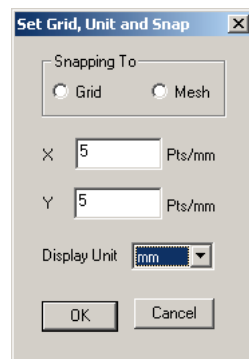
► **To set the display unit:**

1. Click Tools:Options:Grid and Unit from the menu bar to open the Set Grid, Unit and Snap dialog box.
2. Select the preferred display unit in the dialog box: um, mm, cm, m, mil or inch. The unit selected will appear as the default unit in all applicable windows and dialog boxes of the program.

**NOTE:**

The Display Unit last selected in the Set Grid, Unit and Snap dialog box will be saved in the program's environment files and read as the default for each subsequent use of SPDGEN.

3. Click OK.



► **To establish a drawing grid:**

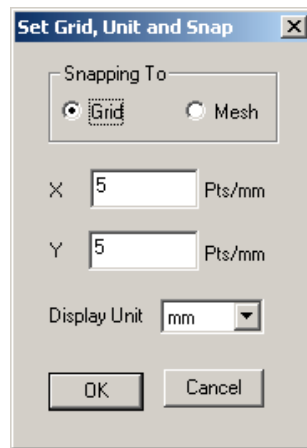
1. Click Tools: Options: Grid and Unit to open the Set Grid dialog box.
2. Enter the number of points along X and Y direction.
3. Click OK.

**NOTE:**

You make the grid visible by checking the View:Show Grid on the main menu.

► **To turn on the “snapping to” feature:**

1. Click Tools: Options: Grid and Unit to open the Set Grid dialog box.



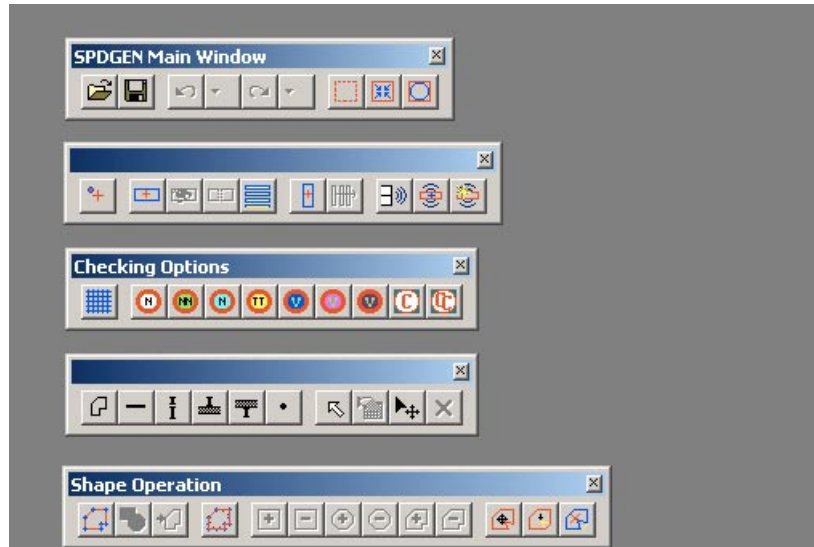
2. Click the button in front of the Grid or Mesh option.
3. Click OK.

**NOTE:**

If you choose the Snapping To “Grid” option to help you place objects precisely, you may need to adjust the positions of the objects placed later due to the numerical discretization error and the mesh. On the other hand, selecting the Snapping To “Mesh” option can help you to avoid discretization errors at the outset. For more information on discretization errors, refer to [“Establishing the numerical mesh” on page 6-2](#).

## Using Drawing Tools

The package drawing tools are grouped into 5 categories: Node Tools, Shape Tools, Via Tools, Trace Tools (including coupled traces) and Error Checking. Specialized toolbars provide convenient access to the graphical editing features. Select View:Toolbar to show/hide the toolbars.



### NOTE:

When you select a tool in one of those 5 groups, that tool will continue in the “on” mode until you select another tool. You can click the right mouse button to exit from the selected tool and get back to the non-drawing mode.

### Node Tools

The end locations of traces and vias are defined as package nodes. The node tools are for controlling the positions of traces and vias. To move a via or one end of a trace, you actually move the node of the via or the node at the end of the trace. Detailed instruction on how to use node tools can be found [“Using Node Tools” on page 4-25](#).

### Shape Tools

Three kinds of shape tools are provided with both add and cut features — they are: box, circle and polygon. The shape tools can be applied to plane layers or patches on signal layers. See the section [“Working on Shapes” on page 4-29](#).

### Via Tools

A via provides interconnection between layers in the vertical direction. Detailed instructions on how to use via tools can be found in the section [“Working on Vias” on page 4-68](#).

### Trace Tools

Traces represent routed electrical connections. For detailed instructions on the operation of trace tools, please refer to [“Working on Traces” on page 4-91](#).

### Error Checking

Some drawing errors can be hard to find. For example, you may think (from appearance) a trace is properly connected to a via — but actually, the nodes of the trace and the via are not properly connected. Use the error checking tools of the software, to assist you to find these “drawing errors”. For detailed instructions on error checking, please refer to [“Checking Drawing Errors” on page 4-99](#).

### Using Node Tools

Node tools are typically used to fine tune the locations of vias and traces. To change the location of a via or one end of a trace, you may select the upper or lower node of the via or a node of a trace, then move that selected node.

#### NOTE:

Before doing any operations on a node, the layer on which the node resides must be activated first.



Select Node button

#### ► To select a node by mouse:

1. Click the *Select Node* button on the tool bar.
2. Move the cursor to the node to be selected and click the right mouse button. The node will be momentarily highlighted by a red dot if the node is selected.
3. Observe the popup menu that is displayed. It provides a Properties icon and reports the package and name of the object (for example, Package1.Node01). Be careful that you have not inadvertently selected the trace object.

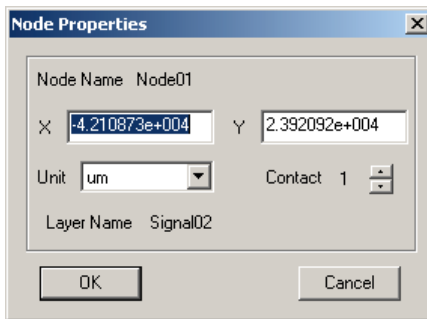


Properties icon

4. Click the Properties icon to display the Node properties dialog box for “Package1.Node01”.



Highly magnified view of the end of a trace. Right mouse button click on the end of the trace to select the node.



**NOTE:**

When the *Select Node* button is activated, the x and y coordinates of the mouse cursor is shown at the lower right corner of the screen.



Property button

► **To move a node by entering the coordinates:**

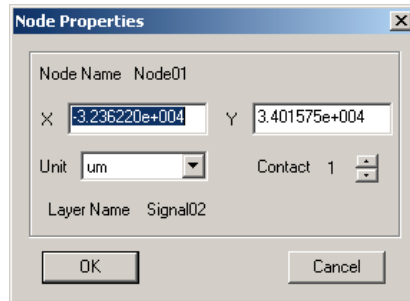
1. After selecting the node, click the *Property* button on the tool bar. The Node Properties dialog box appears.

**NOTE:**

An alternative way is, after selecting the node, click the right mouse button and a flying menu will pop out. Select the *Property* item in the flying menu, the Node Properties dialog box will appear.



2. The Node Properties dialog box shows the current coordinates of the selected node.




3. Enter the x and y coordinates of the new location of the node.
4. Click the OK button to exit the Node Properties dialog box.



Select Node Button

► **To merge two nodes:**

1. Click the *Select Node* button on the tool bar. If necessary, refer to [“Selecting Objects” on page 4-21](#).
2. Select one node by clicking the left mouse button at the node. The node selected will be highlighted by a red dot.
3. Hold down the left mouse button and drag the cursor towards the other node while pressing down the Ctrl key. When the shape of the cursor transforms to , release the left mouse button.

**NOTE:**

After merging two nodes, the objects which terminate at the nodes will be connected. In this manner you can connect two unconnected traces or connect a trace to a via.

As you are placing vias and traces, **it is extremely important to ensure the desired metal connections are actually made. Undesired metal connections or unexpected disconnections may lead to erroneous simulation results.**

► **To alter node “contact” states:**

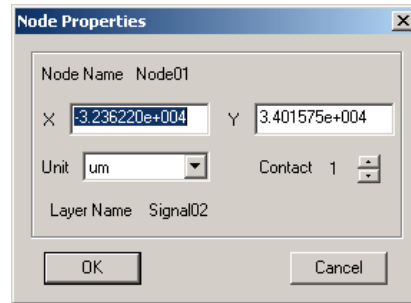
Sometimes nodes reside on metal shapes. Nodes can have two contact states:

- 1 - The node has electric contact with the metal shape.
- 0 - The node has NO electric contact with the metal shape.



Property button

1. After selecting the node, click the *Property* button on the tool bar. The Node Properties dialog box appears.
2. The Node Properties dialog box shows the current state of contact of the selected node.

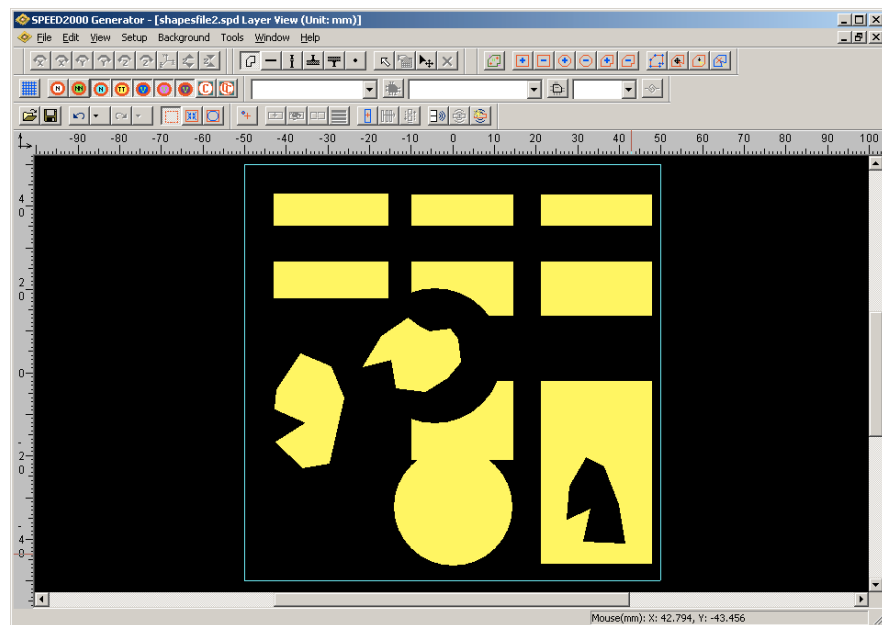


3. Click the arrow buttons to toggle between 1 and 0.
4. Click the *OK* button to exit the Node Properties dialog box.

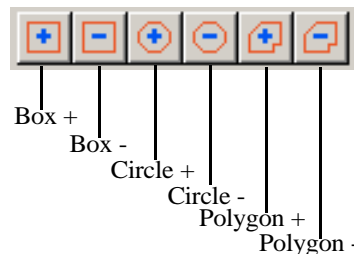
### Working on Shapes

Shapes enable you to model metal geometries which affect the package's performance. Shapes are typically placed on plane layers — however, they may also be present on signal layers. Shapes on signal layers are referred to as “patches”. Primitive shape components are boxes, circles and polygons. From these primitives you can create the most complex shape needed. A collection of shape components comprises a **shape definition**. Each layer can have only one shape definition assigned to it. For operational details refer to [“To create a new Shape Definition:” on page 4-32](#), [“To use a new shape definition on a signal layer:” on page 4-35](#), [“To select a shape definition for a plane layer:” on page 4-38](#), [“To select a shape definition for a patch:” on page 4-38](#) and [“To delete a shape definition:” on page 4-39](#).

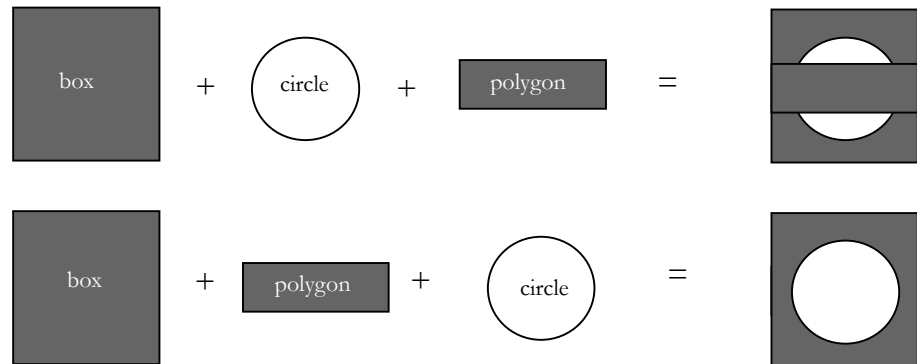
### Positive and Negative Shape Objects



There are six shape related buttons on the tool bar: Box + permits you to draw a positive box shape. Box - permits you to draw a negative box shape. Circle + permits you to draw a positive Circle shape. Circle - permits you to draw a negative Circle shape. Polygon + permits you to draw a positive Polygon shape. Polygon - permits you to draw a negative Polygon shape.



When viewing the shapes on the screen, the positive portion depicts adding (or depositing) a piece of metal to a layer, whereas the negative portion depicts cutting out (or subtracting) a piece of metal from the layer. When making a composite shape on a layer, the order of adding the positive (+) and the negative (-) shapes is important. Shapes added later appear in front of shapes added earlier and serve to effect the final shape. Please review the following for an example:



For operational details, refer to [“To draw a box \(or add a box component to a shape definition\):” on page 4-43](#), [“To draw a circle \(or add a circle component to a shape definition\):” on page 4-44](#) and [“To draw a polygon \(or add a polygon component to a shape definition\):” on page 4-45](#).

### Holes and Voids

In working with shapes, **you do not need to draw via holes**. The effects of via holes are taken care of separately. You **do** need to identify **special voids** that you want to be processed specially in the file. For detailed information, please refer to [“Special Voids — Dog-Leg Holes and Small Holes” on page 4-63](#)

### Shape (and trace) Process function

Often the model components need to be “processed” to eliminate inefficient graphic artifacts — such as overlapping polygons (that need to be shape unionized), traces that overlap positive shapes (patches) on the signal layer and traces that cross split planes. The *Shape Process* button, available on the Shape Toolbar bar, provides a full pre-conditioning function: it clips traces, unionizes shapes, highlights voids (that meet the established size and content criteria for *special voids*) and segments traces that cross split planes. For more information, please refer to [“To unionize shapes \(only\):” on page 4-55](#), [“To clip traces and unionize shapes:” on page 4-59](#), [“Shape \(and trace\) Process function” on page 4-30](#), [“To establish special void size criteria:” on page 4-64](#) and [“Traces Over “Split Planes”” on page 4-97](#).



Shape Process button

**NOTE:**

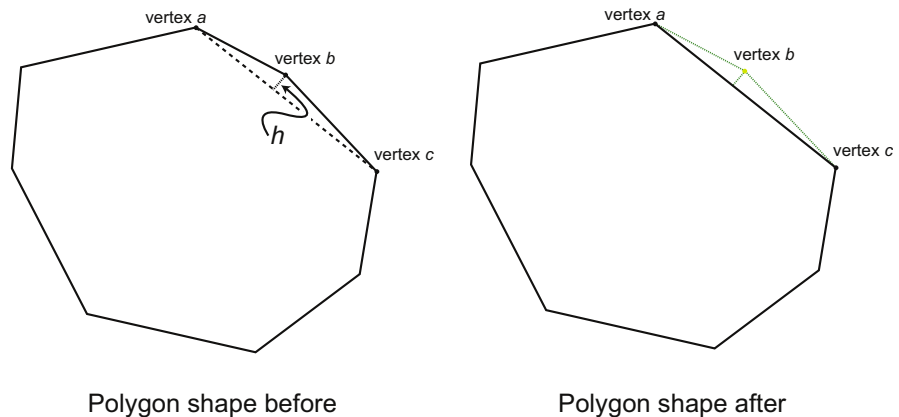
Do not avoid full pre-conditioning of the .spd file prior to simulation or erroneous simulation results may occur. Proper/full pre-conditioning of the .spd file, consists of:

- unionizing shapes
- clipping traces
- identifying any "special voids" (per the User's criteria)
- and segmenting traces (with the Trace Over Split Plane function). The *Shape Process* button, available on the Shape Toolbar, performs all of these functions.

**Polygon Smoothing/Simplifying**

The Polygon Smoothing/Simplifying function makes simulations faster by smoothing out the “sharp edges” of polygon shapes, using the Polygon Smoothing Algorithm (PSA). The PSA reduces the number of vertices in the polygons representing shapes — thereby “simplifying” them. The user can set a threshold value as shown below. A default value is dynamically supplied based upon the dimensions of the package when it is loaded.

If the Polygon simplifying function is activated, and if distance  $h$  is less than the "threshold" value, then vertex  $b$  of the polygon (and its associated line segments) will be removed.

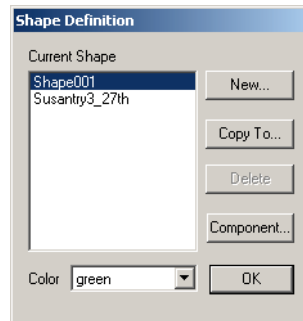
**NOTE:**

Care must be taken to ensure the Polygon Smoothing Algorithm does not change your polygon shapes undesirably — use of the default threshold value is recommended.

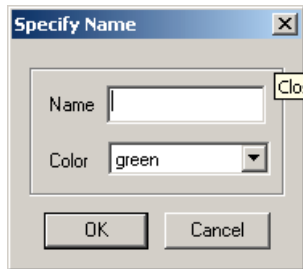
### Working with Shape Definitions

► **To create a new Shape Definition:**

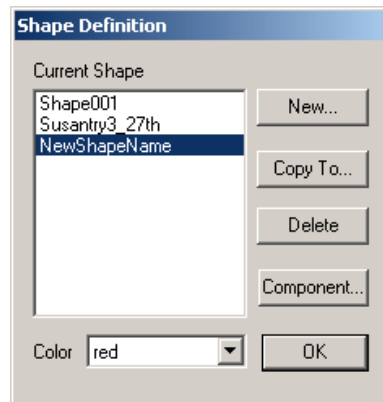
1. With the Layer View window active, select Edit:Shape:Shape Definition, the Shape Definition dialog box appears.



2. Click the *New* button, the Specify Name dialog box appears. Enter a name for the new shape definition and select a default color from the drop down list.



3. Click the *OK* button when you are done, the Shape Definition dialog box will be updated with the new shape name entered.



4. Click *OK* to dismiss the dialog box.

**NOTE:**

You may also click the *Shape Definition* button on the Plane and Signal Properties dialog box.

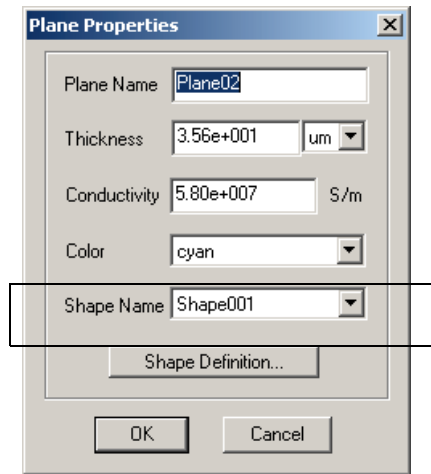
► **To use a new shape definition on a plane layer:**

Each plane layer can be associated with only one shape definition. The shape definition assigned — and the box, circle, polygon and special void components contained within that definition — will be associated with that plane layer until you select another shape definition or edit the assigned shape definition. A “new” shape definition does not contain any shape components, they must be added.

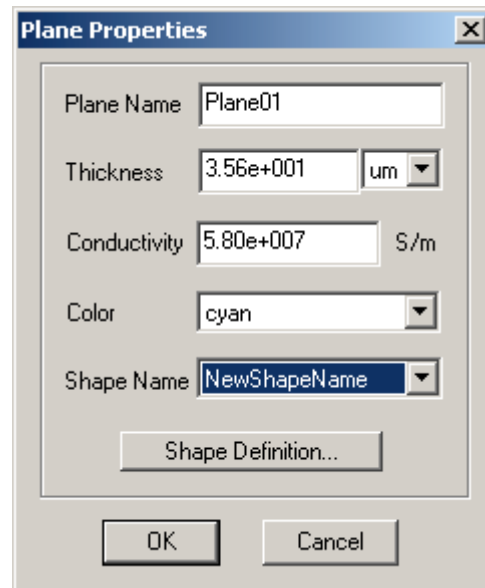
1. With the Layer View window active, select *Edit:Stack Up* from the menu bar, the Stack Up dialog box appears.



2. Double-click on the plane layer of interest, the Plane properties dialog box appears.



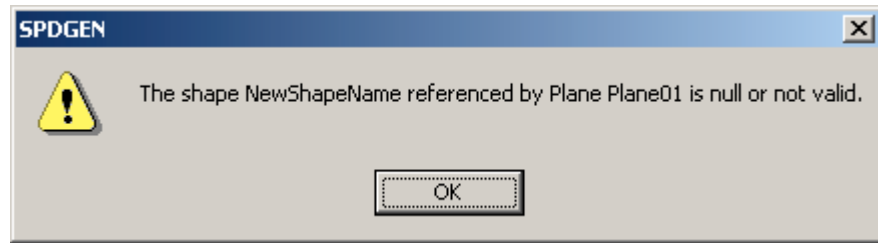
3. Select the new shape definition name from the drop down listing. Recall, you must have already created a Shape Definition to use it at this point. If necessary, refer to [“To create a new Shape Definition:” on page 4-32.](#)



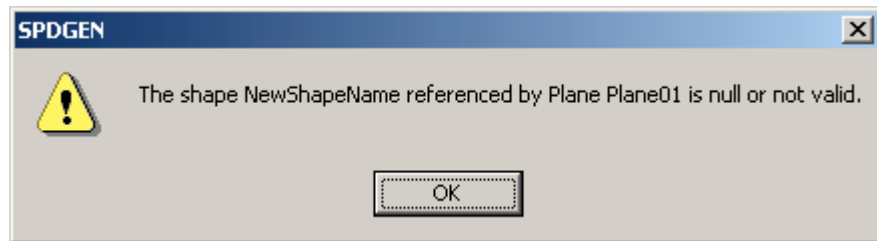
4. Click the OK button on the Plane Properties dialog box. A message appears to inform you that the Shape Definition that you are assigning to this layer is empty.



Click *OK* to dismiss the message. The new shape definition (even though empty) is assigned to the plane layer.



5. Click *OK* on the Stack Up dialog box to dismiss it. Again, a message appears to inform you that the Shape Definition that you are assigning to this layer is empty. Again, click *OK* to dismiss the message. The new shape definition (even though empty) is assigned to the plane layer.

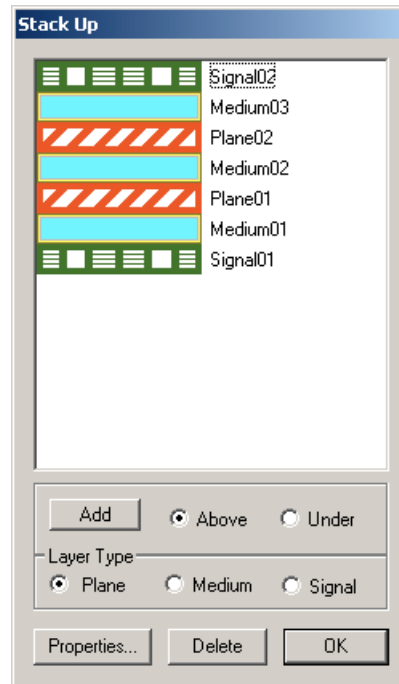


6. With the Layer View window active, select View:Layer Selection, the Layer Selection dialog box appears.
7. Set the plane that you have just assigned the new shape definition to as the active plane. Toolbar:Shape should be visible with all buttons active, select View: Toolbar-Shape if necessary.
8. At this point you may add shape components as you deem appropriate, refer to detailed instructions: [“To draw a box \(or add a box component to a shape definition\):” on page 4-43](#), [“To draw a circle \(or add a circle component to a shape definition\):” on page 4-44](#) and [“To draw a polygon \(or add a polygon component to a shape definition\):” on page 4-45](#).

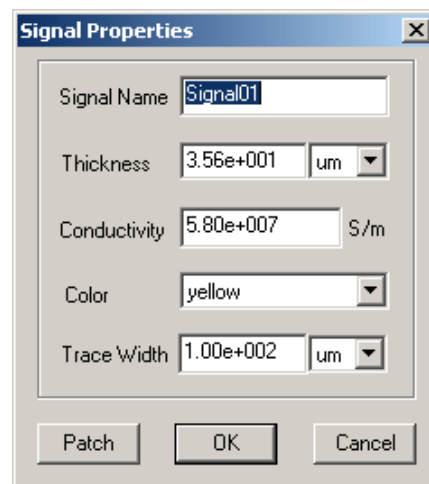
► **To use a new shape definition on a signal layer:**

Each signal layer can be associated with only one shape definition. The Shape Definition assigned — and the box, circle, polygon and special void components contained within that definition — will be associated with that signal layer until you select another shape definition or edit the assigned shape definition. A “new” shape definition does not contain any shape components (it is a “null” shape definition), shapes must be added.

1. With the Layer View window active, select Edit:Stack Up from the menu bar, the Stack Up dialog box appears.

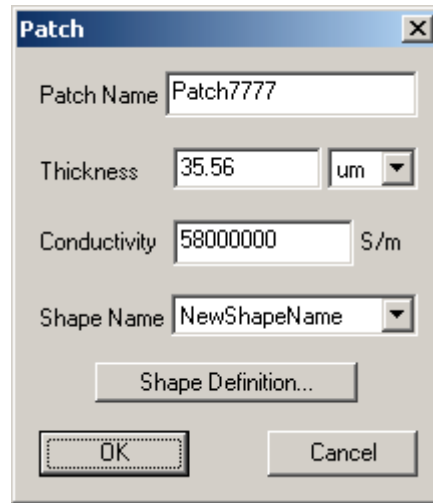


2. Double-click on the signal layer of interest, the Signal properties dialog box appears. Whereas, shape definitions can be added directly to plane layers, signal layers can only contain “patches” — which, in turn, contain the shape definition, so, an additional step is necessary to assign a name for the patch.

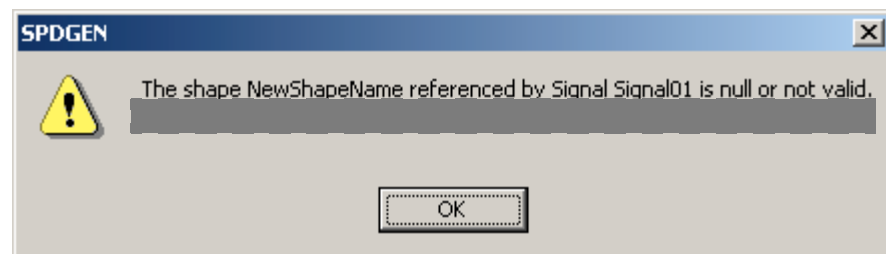


3. Click the *Patch* button, the Patch dialog box appears. Assign a name for the patch, it must begin with P-a-t-c-h (as shown). Select the new shape definition name from the drop down listing. Recall, you must have already created a Shape Defi-

inition to use it at this point. If necessary, refer to [“To create a new Shape Definition:” on page 4-32.](#)



4. Click the *OK* button on the Patch dialog box to dismiss it. Click the *OK* button on the Signal Properties dialog box to dismiss it. Click the *OK* button on the Stack Up dialog box to dismiss it. A message appears to inform you that the Shape Definition that you are assigning to this layer is empty. Click *OK* to dismiss the message. The new shape definition (even though empty) is assigned to the signal layer.



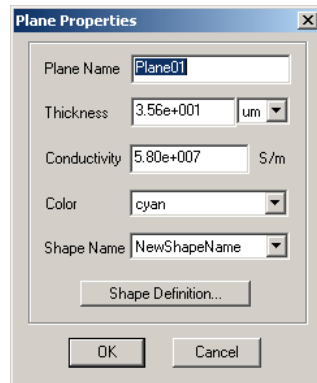
5. With the Layer View window active, select View:Layer Selection, the Layer Selection box appears.
6. Set the signal layer that you have just assigned the new shape definition to as the active plane. Toolbar:Shape should be visible with all buttons active, select View: Toolbar-Shape if necessary.
7. At this point you may add shape components as you deem appropriate, refer to detailed instructions: [“To draw a box \(or add a box component to a shape definition\):” on page 4-43](#), [“To draw a circle \(or add a circle component to a shape definition\):” on page 4-44](#) and [“To draw a polygon \(or add a polygon component to a shape definition\):” on page 4-45](#).

► **To select a shape definition for a plane layer:**

**NOTE:**

Several layers can share one shape definition — however, when you edit the shape definition, all layers which use that definition are affected.

1. Click Edit:Stack Up from the menu bar. The Stack Up dialog box appears.
2. In the Stack-Up dialog box, double click on the name of the plane layer. The Plane Properties dialog box appears.



3. In the Plane Properties dialog box, go to the Shape Name list box and select the shape for this plane.
4. Click the OK button to exit the Plane Properties dialog box.

**NOTE:**

To view the shape of a plane, first make the plane as the only visible layer in the Layer Selection dialog box. Then check the View: Show Shape option on the menu bar. For a new shape definition refer to [“To use a new shape definition on a plane layer:” on page 4-33](#).

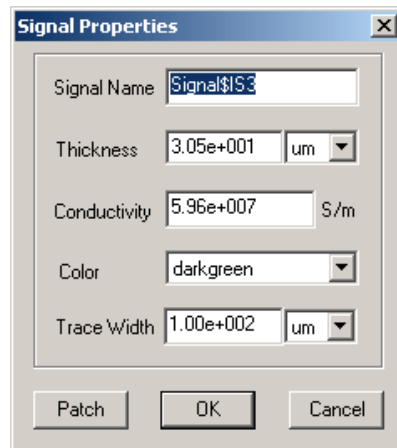
► **To select a shape definition for a patch:**

**NOTE:**

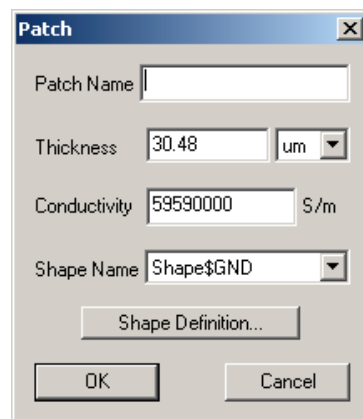
Several layers can share one shape definition — however, when you edit the shape definition, all layers which use that definition are affected.

1. Click Edit:Stack Up from the menu bar. The Stack Up dialog box appears.

2. In the Stack-Up dialog box, double click on the name of the signal layer. The Signal Properties dialog box appears.



3. Click the *Patch* button. The Patch dialog box appears.



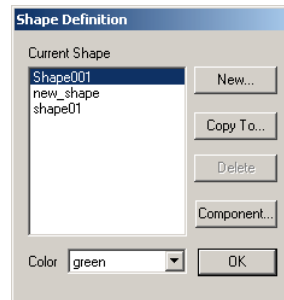
4. Select the Shape Definition in the Shape Name pull down list box. Give the patch a name, if it is absent, in the Patch Name field. The name must start with "Patch".
5. Click the *OK* button to exit the Patch dialog box.

#### NOTE:

To view the shape of a patch, first make the signal layer as the only visible layer in the Layer Selection dialog box. Then check the View: Show Shape option on the menu bar. For a new shape definition refer to ["To use a new shape definition on a signal layer:" on page 4-35](#).

#### ► To delete a shape definition:

1. Click Edit:Shape:Shape Definition from the menu bar. The Shape Definition dialog box appears.

**NOTE:**

You can also open the Shape Definition dialog box by clicking the *Shape Definition* button inside the Plane Properties dialog box or the Patch Properties dialog box (available from the Signal Properties dialog box).

2. Highlight the name of the shape definition in the Shape Definition dialog box.

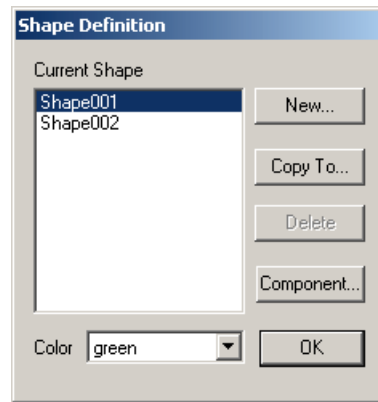
**NOTE:**

**If the shape definition is already assigned to a plane or signal layer, you cannot delete the shape definition.** The *Delete* button is automatically grayed if the highlighted shape definition has been assigned to at least one plane or signal layer. For more information, refer to [“To use a new shape definition on a plane layer:” on page 4-33](#).

3. Click the *Delete* button on the right-hand side of the Shape Definition dialog box.
4. Click the *OK* button to exit the Shape Definition dialog box.

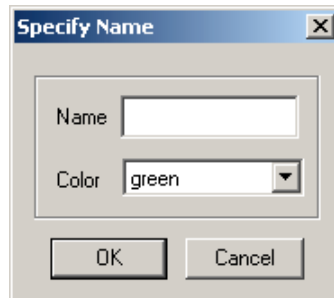
► **To use the “Copy To” button for a shape definition:**

1. Click Edit:Shape:Shape Definition from the menu bar. The Shape Definition dialog box appears.

**NOTE:**

You can also open the Shape Definition dialog box by clicking the *Shape Definition* button inside the Plane Properties dialog box or the Patch Properties dialog box (available from the Signal Properties dialog box).

2. Observe the Current Shape definition highlighted. This is the Shape Definition that will be “copied to” the new name specified.
3. Click the *Copy To* button in the Shape Definition dialog box, the Specify Name dialog box appears.



4. Enter the name of the new shape definition, then click the *OK* button to exit the Specify Name dialog box.

**NOTE:**

The new shape definition created by copying from an existing shape definition inherits all the graphics objects of the original shape definition. However, if the contents of the original shape definition are later changed, the contents of the new shape definition are **not** changed accordingly.

5. Click the *OK* button to exit the Shape Definition dialog box.

**NOTE:**

To modify the contents of a shape definition, you have to first assign the shape definition to a plane or signal layer, then activate the layer and do the editing. **Once the contents of the shape definition are changed, shapes of all the layers using that shape definition are changed.**



### **Drawing Boxes**

► **To draw a box (or add a box component to a shape definition):**

1. Make certain the layer that you want to add the box to is active in the layer view window, the Show Shapes function is active and the shape definition that you want the box to become a component of is assigned to the layer. If necessary, refer to [“Viewing Layers and the Layer Selection Dialog Box” on page 4-5](#).
2. Click the *Box+* button on the tool bar to add a positive box shape or the *Box-* button on the tool bar to add a negative box shape. The cursor appears as a + as it moves into the Layer View window.

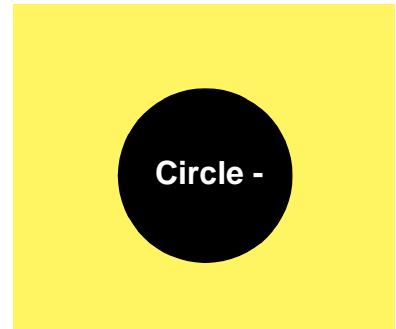
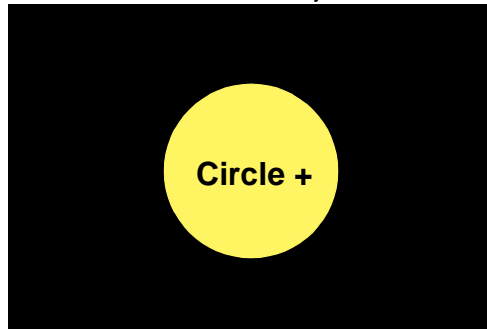


3. Move the cursor to the location of the upper-left corner of the box to be added, and click the left mouse button.
4. Hold down the left mouse button and drag the cursor to the location of the lower-right corner of the box. The box outline should appear on the screen. Release the left mouse button to complete the adding of the box.

### **Drawing Circles**

► **To draw a circle (or add a circle component to a shape definition):**

1. Make certain the layer that you want to add the circle to is active in the layer view window, the Show Shapes function is active and the shape definition that you want the circle to become a component of is assigned to the layer. If necessary, refer to [“Viewing Layers and the Layer Selection Dialog Box” on page 4-5](#).
2. Click the *Circle+* button on the tool bar to add a positive circle shape or the *Circle-* button on the tool bar to add a negative circle shape. The cursor appears as a + as it moves into the Layer View window.

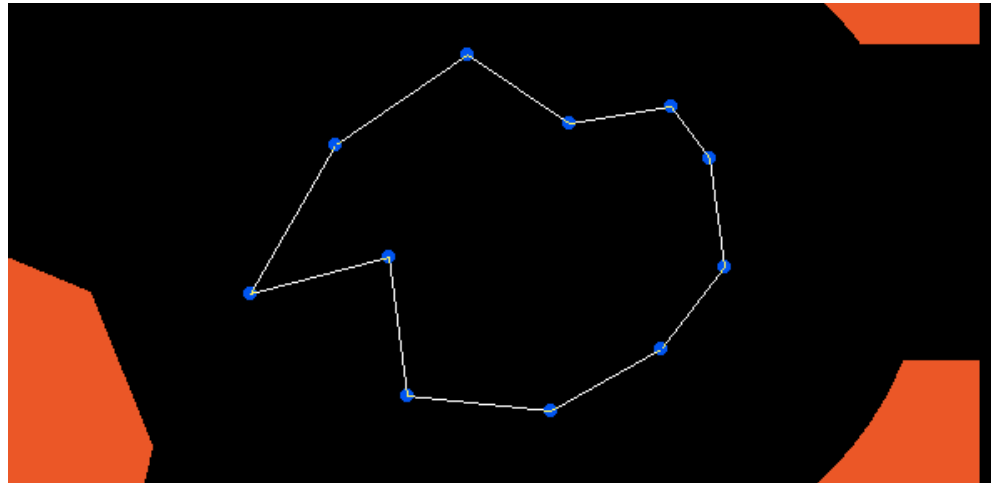


3. Move the cursor to the location of the center of the circle to be added, and click the left mouse button.
4. Hold down the left mouse button and drag the cursor away from the center of the circle. The circle outline should appear on the screen. Release the left mouse button to complete the adding of the circle.

### Drawing Polygons

► **To draw a polygon (or add a polygon component to a shape definition):**

1. Make certain the layer that you want to add the polygon to is active in the layer view window, the Show Shapes function is active and the shape definition that you want the polygon to become a component of is assigned to the layer. If necessary, refer to [“Viewing Layers and the Layer Selection Dialog Box” on page 4-5](#).
2. Click the *Polygon +* button on the tool bar to add a positive polygon shape or the *Polygon -* button on the tool bar to add a negative polygon shape. The cursor appears as a + as it moves into the Layer View window.
3. Move the cursor to the first vertex of the polygon and click the left mouse button once. A blue dot should appear at the place you clicked the mouse.
4. Move the cursor and click at the desired locations for the rest of the vertices of the polygon. You will notice a “rubber band” line following your cursor.



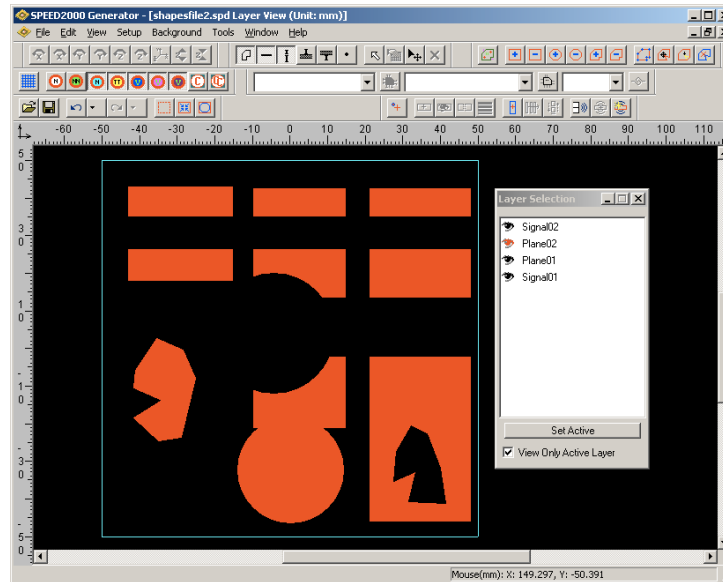
5. After you have clicked all the vertices of the polygon, **double click the left mouse button** to complete the drawing of the polygon.



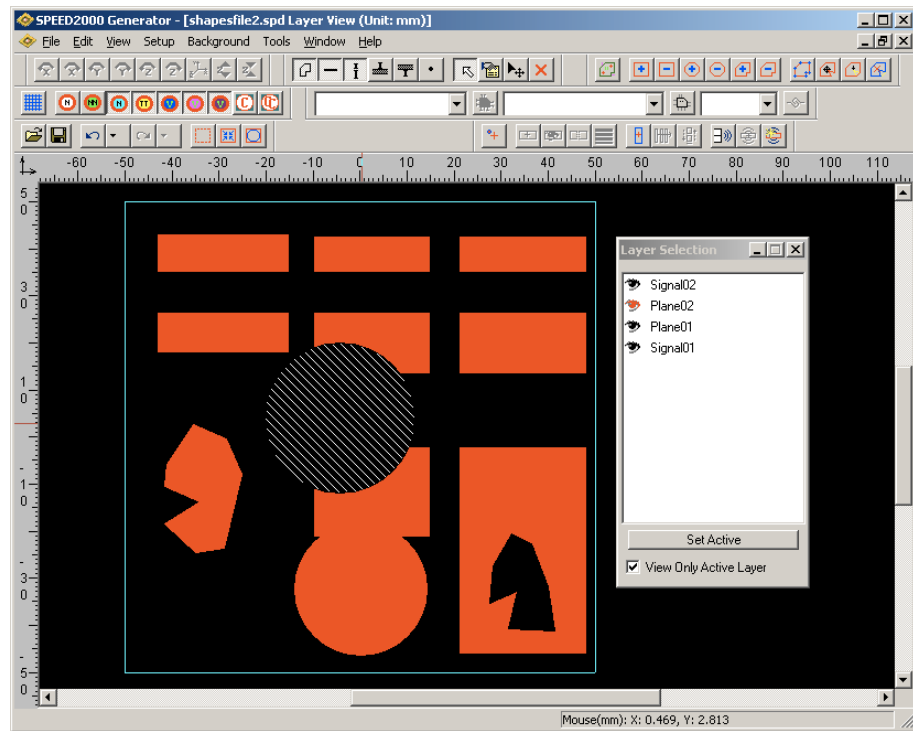
## Deleting Shape Objects

► To delete shape objects (within the Layer View window):

1. Make certain the shape component you want to delete is on the active layer and visible. If necessary, refer to [“Viewing Layers and the Layer Selection Dialog Box”](#) on page 4-5.

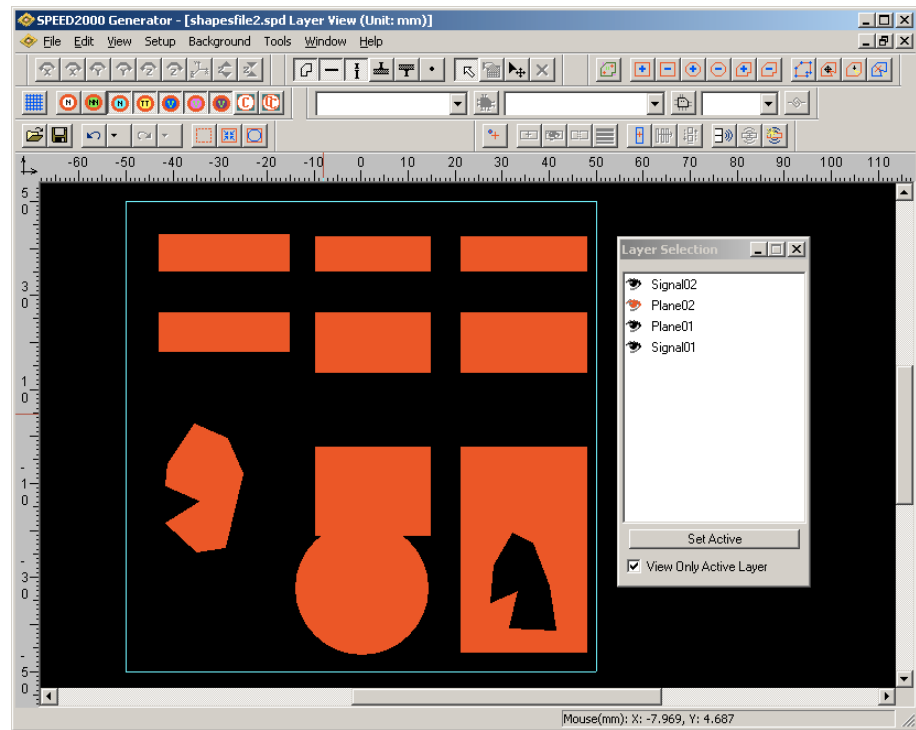


2. Either right-click or use the *Select* button to activate the object of interest.



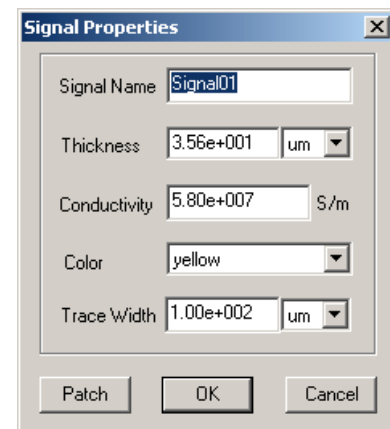
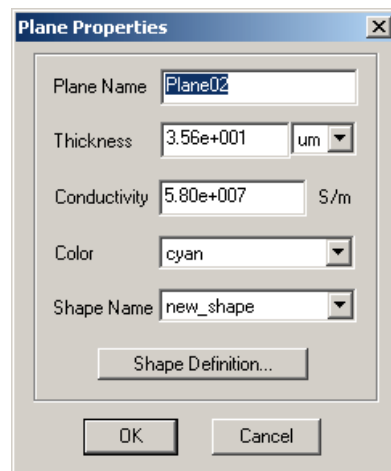


3. Click the *Delete* button (on the Tool bar menu) or press the delete key on the keyboard.

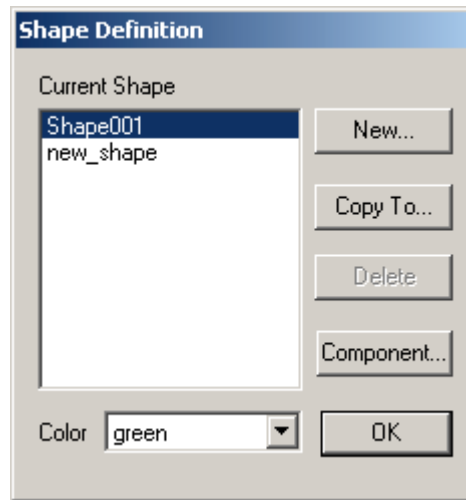


► **To delete shape objects using the Shape Component dialog box:**

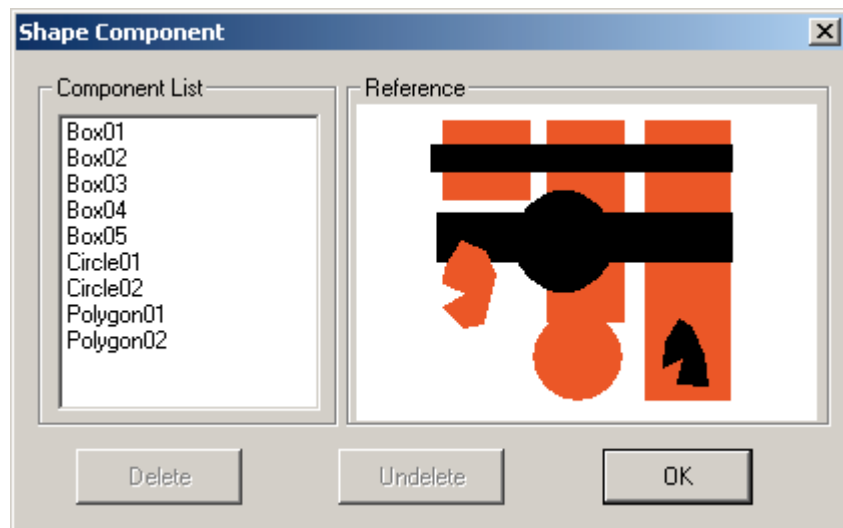
1. Make certain the Layer View window is active.
2. Click Edit:StackUp from the menu, the Edit Stackup dialog box appears.
3. Double-click on any layer that contains the Shape Definition of interest, the Plane or Signal layer Properties dialog box appears.



4. Click the *Shape Definition* button. For the *Plane Properties* dialog box you may just click on the *Shape Definition* button. For the *Signal Properties* dialog box, click the *Patch* button, the *Patch* dialog box appears, then click the *Shape Definition* button. After clicking the *Shape Definition* button, the Shape Definition dialog box appears.

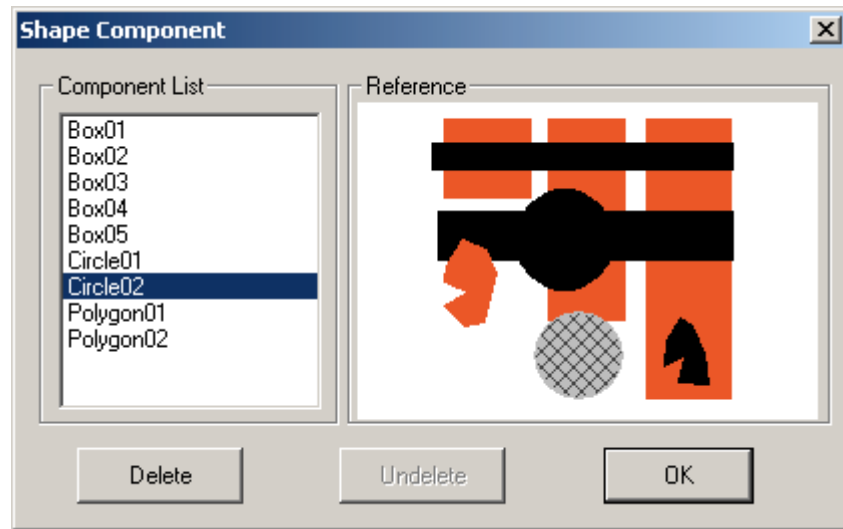


5. Observe the current shape definitions defined for the package. Click on the shape definition of interest, it will be highlighted. Click on the *Component* button, the Shape Component dialog box appears.



6. Observe the Component List and click on the component that you want to delete. The highlighted object will be displayed in the Reference area and the

*Delete* button will become active. Click the *Delete* button, the object will be deleted.

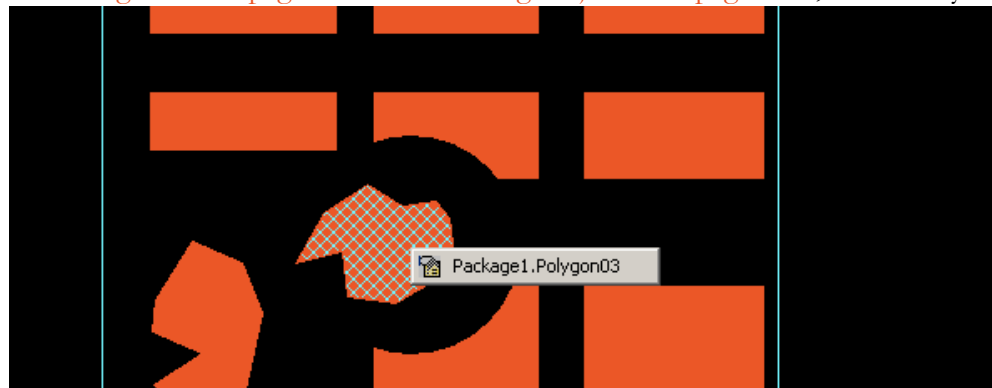


7. Click OK to exit the Shape Component dialog box.

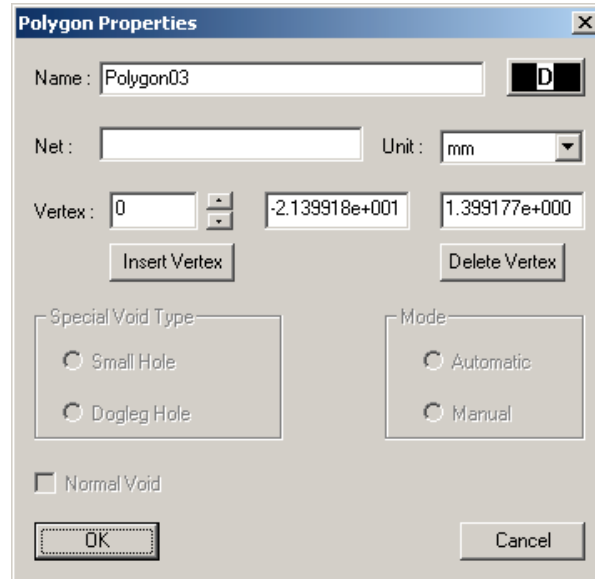
### Viewing and Editing Shape Properties

#### ► To view a shape component's property box:

1. If a layer containing shape components is active and the Show Shapes function is activated, right-clicking the mouse button on any shape component will bring out their property menu. Refer to, [“Viewing Layers and the Layer Selection Dialog Box”](#) on page 4-5 and [“Showing Objects”](#) on page 4-20, if necessary.



2. Click the property menu to view the property dialog box for the component. You can change many attributes of the shape from the shape's Properties dialog box.



3. Click the OK button to enter the new attributes and dismiss the Properties dialog box.

**NOTE:**

Numerous separate discussions detail “How To” use the features of each shape dialog box. Please refer to the [“Table of Contents” on page 1-III](#) and [“Index” on page 1-1](#) for pointers and hyperlinks.



Move button

**Moving and Editing Shape Components**

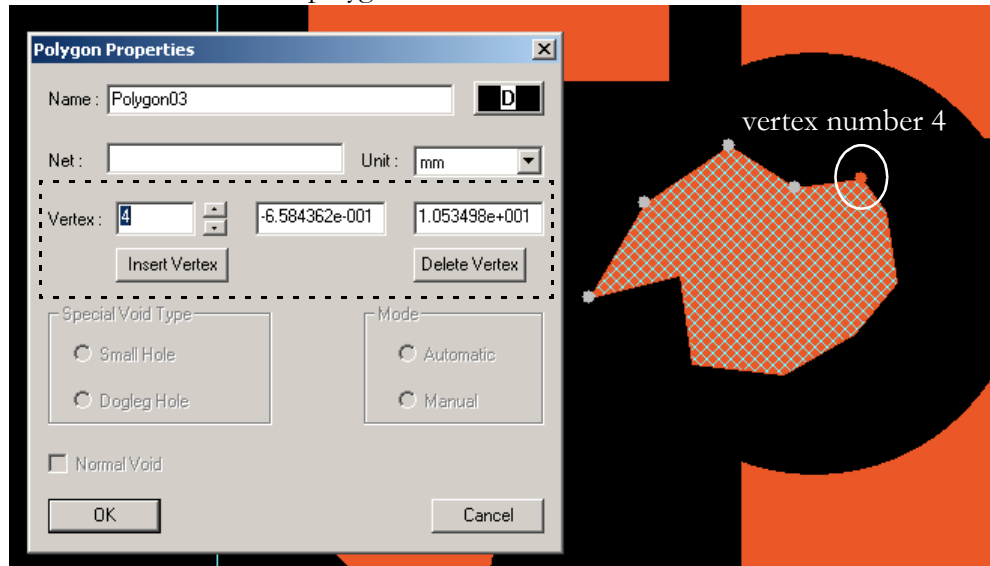
► **To generally move shapes:**

1. Click the *Move* button on the tool bar.
2. Then, click and drag on the shape object of interest to move it.
3. The cursor will remain in the object “move” mode until another button is selected on the tool bar — or the user “right-clicks” in the window work space.

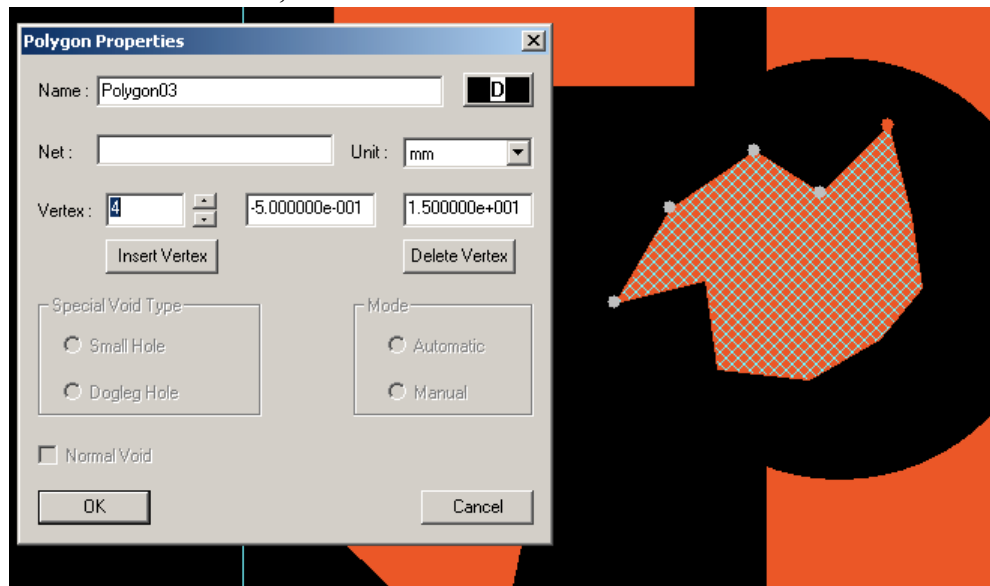


► **To precisely move a vertex of a polygon:**

1. Bring out the property menu of the polygon. Refer to [“To view a shape component’s property box:” on page 4-49.](#)
2. Click on the up and down arrows to select the vertex of interest. Observe how the vertices are highlighted as you click and the specific coordinates are reported for each vertex of the polygon.



3. When the vertex of interest is reached, enter new coordinate information and click the OK button, the vertex will be moved to the new location.





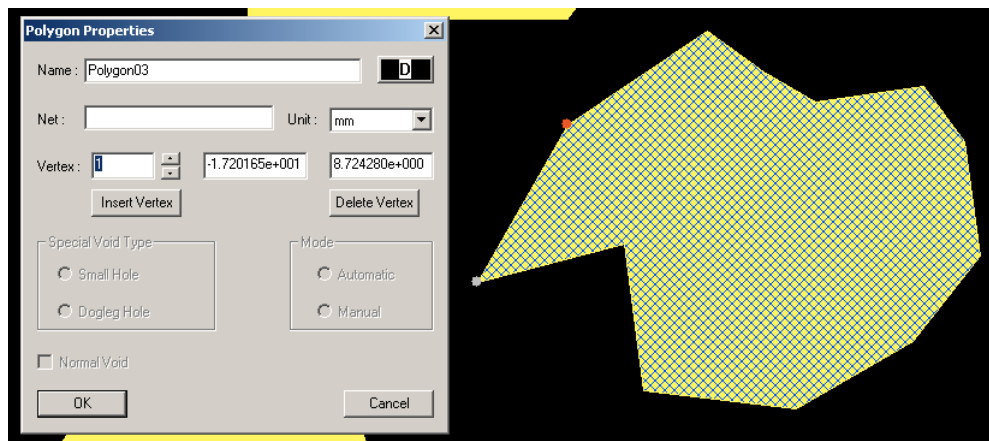
Move Vertex button

► **To generally move a vertex of a polygon:**

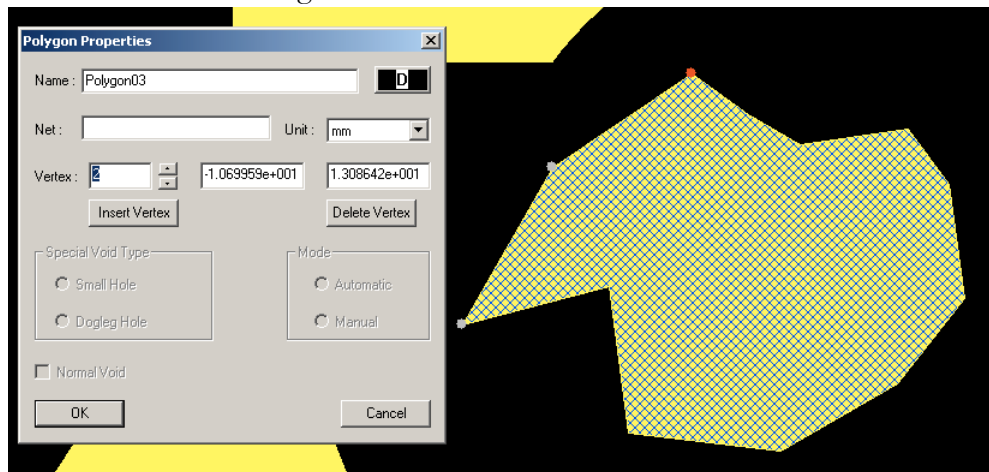
1. Click the *Move Vertex* button on the Shape tool bar.
2. Move the cursor to the vertex to be selected, and click the left mouse button, hold and drag the vertex to a new location. You may then use the left mouse button to select another vertex of the polygon and move it to its desired location.
3. The cursor will remain in the “move vertex” mode until another button is selected on the tool bar — or the user “right-clicks” in the window work space.

► **To precisely add a new vertex to a polygon :**

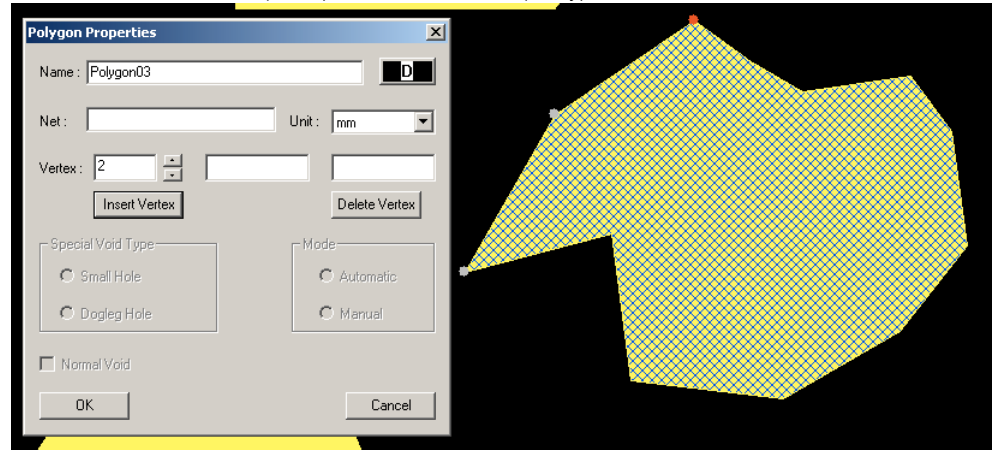
1. Bring out the property menu of the polygon. Refer to [“To view a shape component’s property box:” on page 4-49.](#)
2. Begin by inspecting the polygon vertices to decide where you want to add the new vertex. Click the up and down arrows beside the Vertex field to activate the node vertices. The numbers to the right of the field are the precise x- and y-coordinate values of the vertex.



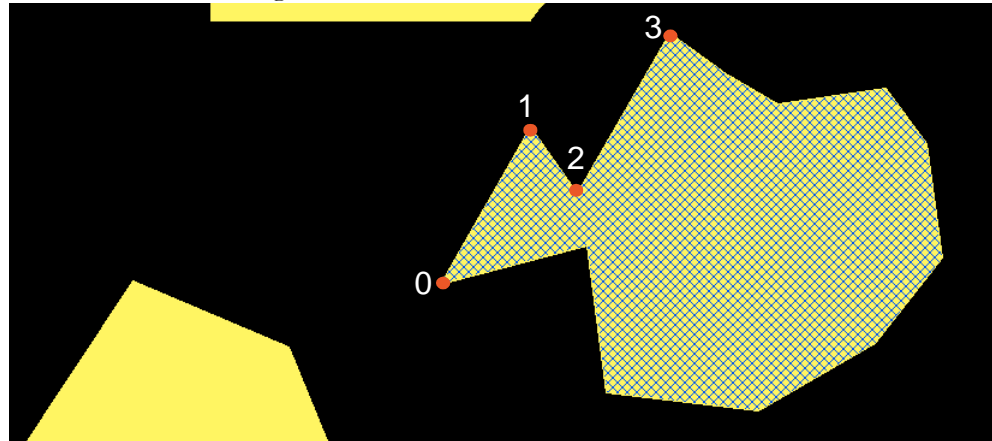
3. When you have activated the node that completes the line segment where you want to add a vertex, stop and press the *Insert Vertex* button — the x- and y-coordinate fields will go blank.



4. Enter the new values for the x- and y-coordinates and press the *OK* button. A new vertex will be added to the polygon — precisely as specified. For example, insert  $-1.5 \text{ e}+001$  (for x) and  $5.7 \text{ e}+000$  (for y).



5. The new numbering of the vertices of interest is shown below.



Insert Vertex button

► **To generally add a new vertex to a polygon:**

1. Click the *Insert Vertex* button on the tool bar.
2. Move the cursor to the polygon edge where the vertex is to be inserted and click the left mouse button. The new vertex will be inserted.
3. The cursor will remain in the “insert vertex” mode until another button is selected on the tool bar — or the user “right-clicks” in the window work space.



Delete Vertex button

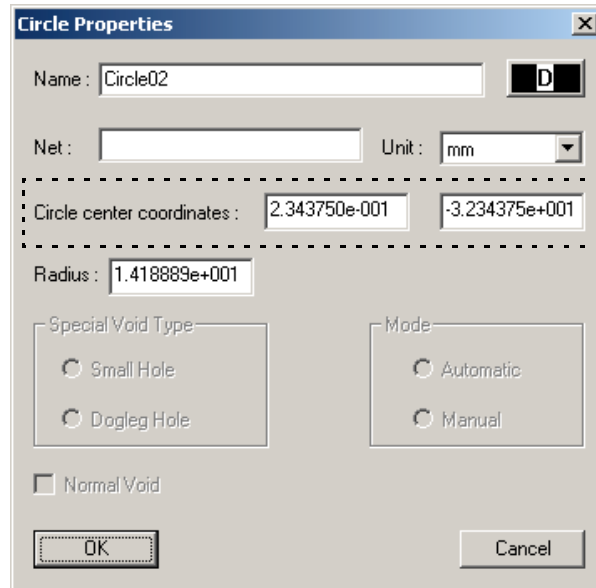
► **To delete a vertex of a polygon:**

1. Click the *Delete Vertex* button on the Shape tool bar.
2. Move the cursor to the vertex to be deleted, and click the left mouse button. The vertex will be deleted. You may then use the left mouse button to select another vertex of the polygon.

3. The cursor will remain in the “delete vertex” mode until another button is selected on the tool bar — or the user “right-clicks” in the window work space.

► **To precisely move a circle:**

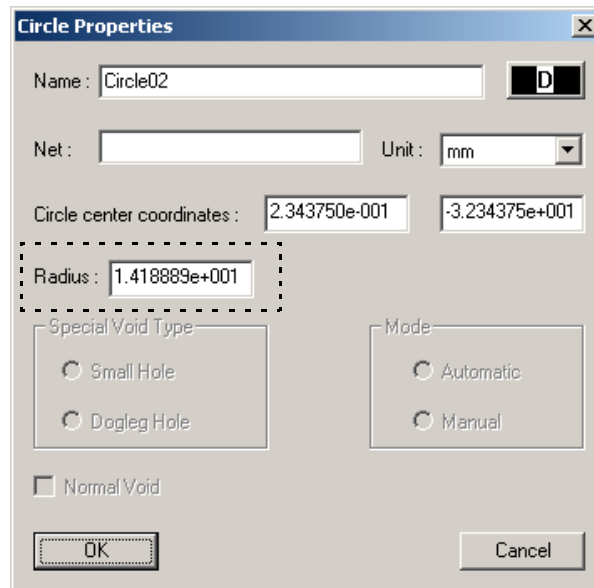
1. Bring out the property menu of the circle. Refer to [“To view a shape component’s property box:” on page 4-49](#).
2. Change the Circle center coordinates as needed.



3. Click OK to enter the new center coordinates and dismiss the property box.

► **To precisely change a circle's size:**

1. Bring out the property menu of the circle. Refer to [“To view a shape component's property box:” on page 4-49](#).
2. Change the Radius as needed.



3. Click OK to enter the new radius and dismiss the property box.

### **Pre-Conditioning the .spd File For Simulation**

#### **NOTE:**

Do not avoid full pre-conditioning of the .spd file prior to simulation or erroneous simulation results may occur. Proper/full pre-conditioning of the .spd file, consists of:

- unionizing shapes
- clipping traces
- identifying any "special voids" (per the User's criteria)
- and segmenting traces (with the Trace Over Split Plane function). The *Shape Process* button, on the Shape Toolbar, performs all of these functions.

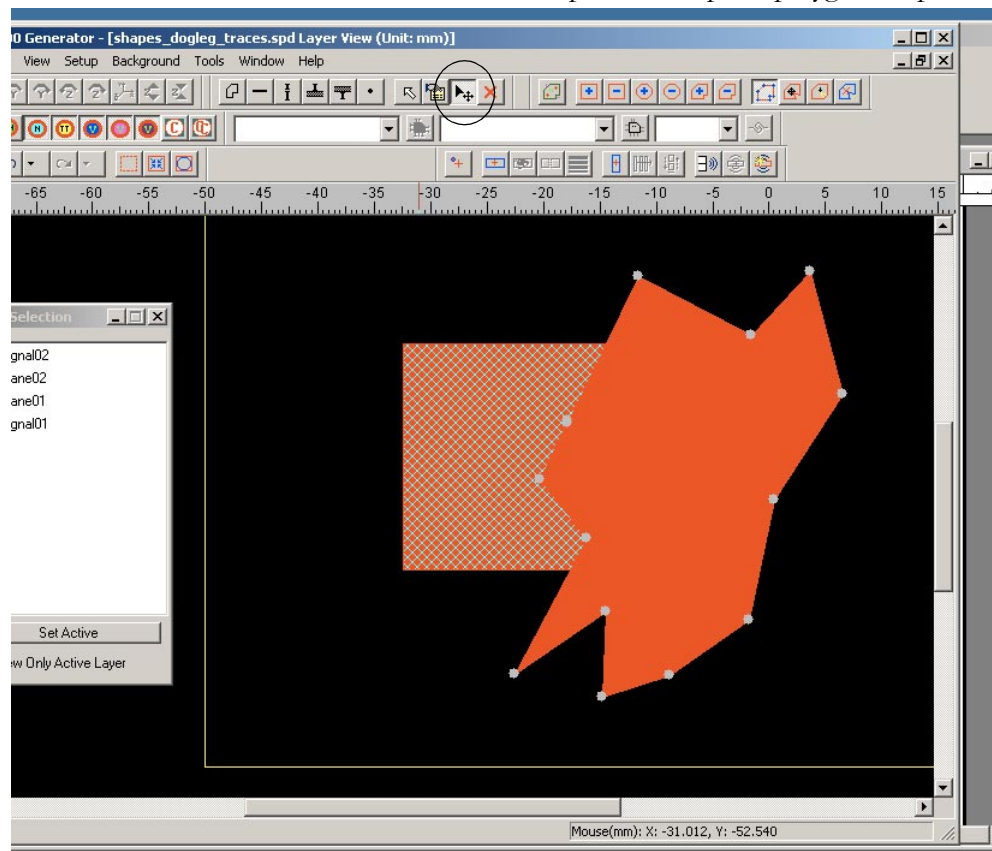
► **To unionize shapes (only):**

Sometimes you just want to unionize shapes — and avoid trace clipping. (Here's how to do it.) The shape unionizing function combines overlapping polygons (or boxes) into one unique polygon. The following steps illustrate the function.

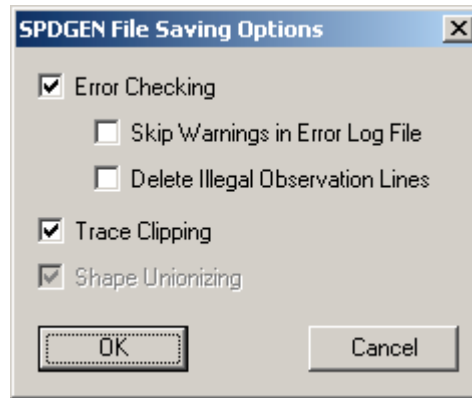
1. Activate a layer with a shape definition. Draw a polygon and a box separately.



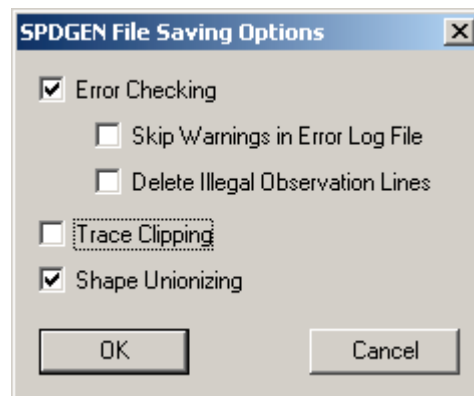
2. Click the *Move* button and move the box shape to overlap the polygon shape.



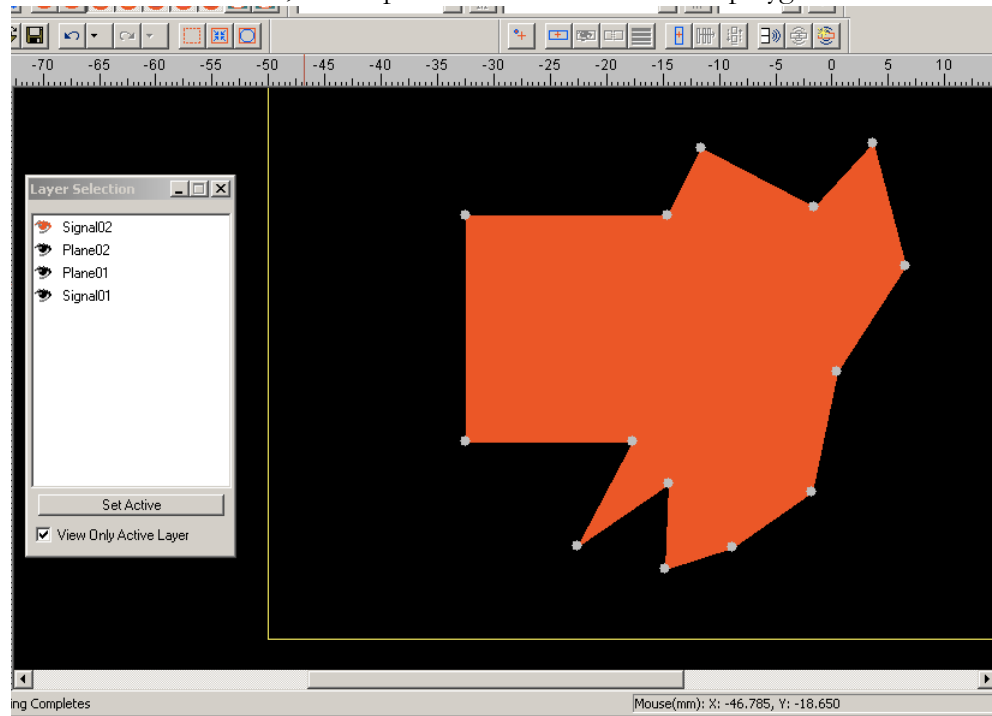
3. Click the File:Save menu option, The SPDGEN File Saving Options dialog box appears.



4. Click the box that precedes the Trace Clipping option, the Shape Unionizing option is highlighted. Click the box that precedes the Shape Unionizing option, if necessary, to place a check mark in the box.



5. Click the *OK* button, the shapes are “unionized” into one polygon.





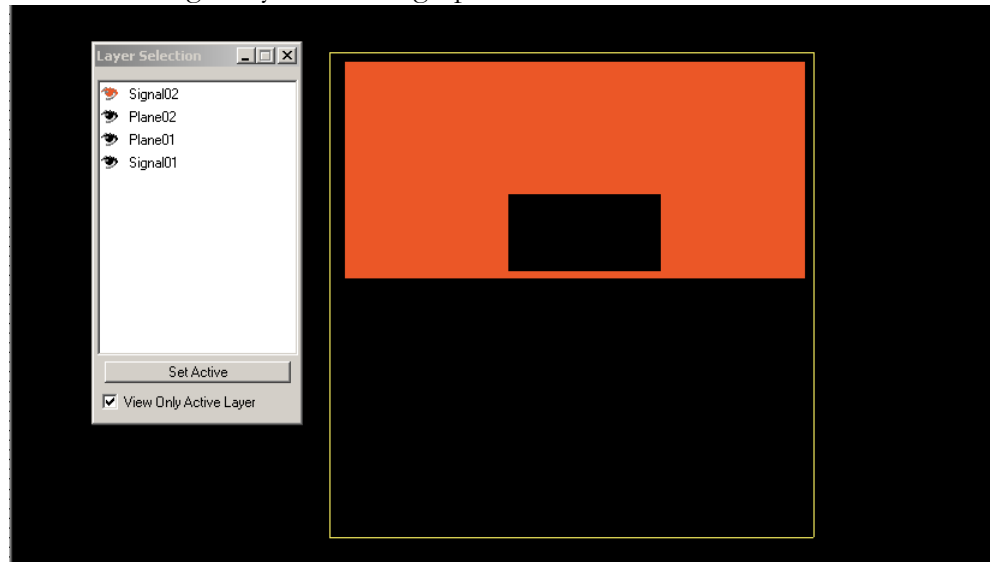
**► To clip traces and unionize shapes:**

Shapes on signal layers are referred to as “patches”. The trace clipping function “clips” traces that overlap positive patch components. The shape unionizing function is run (internally) before the traces are clipped. The following steps demonstrate the functions. To just unionize shapes, refer to [“To unionize shapes \(only\):” on page 4-55](#).

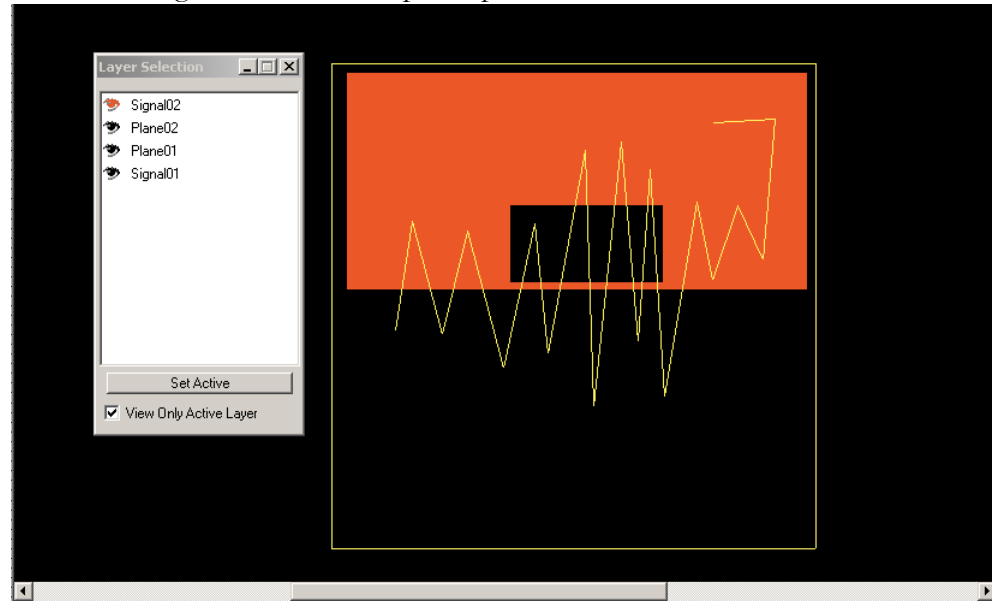
**NOTE:**

One cannot “just” clip traces.

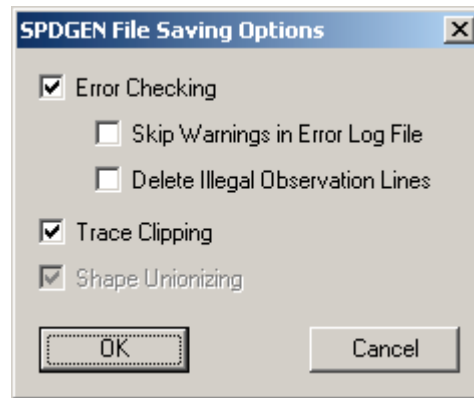
1. Activate a signal layer containing a patch.



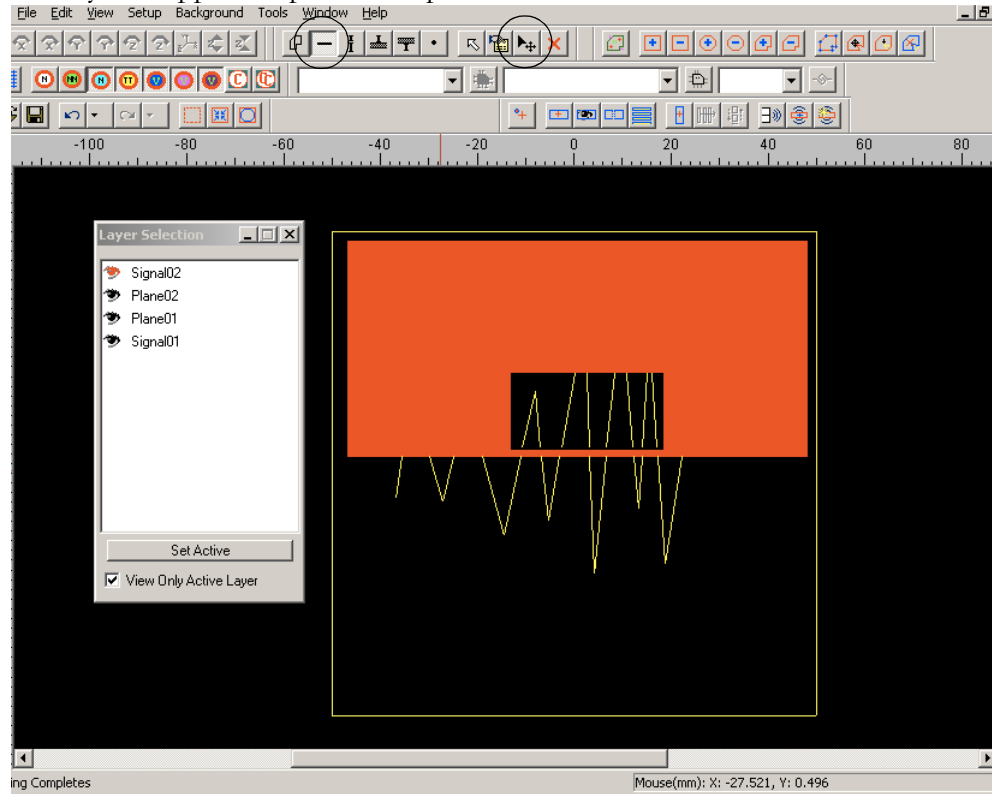
2. Draw a single trace that overlaps the patch.



3. Select File:Save The SPDGEN File Saving Options dialog box appears. Click on the Trace Clipping option to place a checkmark in the box preceding it.
4. Then click the OK button. Traces that overlap the positive shapes on the active layer will be clipped *and the shape unionize function will be run.*



5. Click the *trace select* and *move* buttons and select any remaining trace to observe that the original single trace has now been segmented into many traces where they overlapped the positive shape.



► **To segment and clip traces, unionize shapes, highlight special voids:**

1. With the Layer View window active and the Shape Toolbar visible, click the *Shape Process* button.



Shape Process button

**NOTE:**

Once clicked, the Shape Process function affects the entire package — not just the active layer.

### ***Polygon Smoothing***

► **To smooth polygons in the file:**

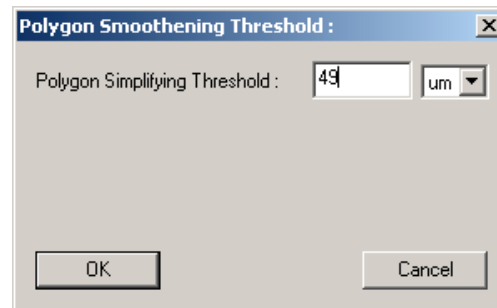
1. With the Layer View window active, select Edit:Shape:Simplify Polygons. All shapes in the .spd file will be simplified using the “default” or a new value entered to the Polygon Smoothing Threshold dialog box.

**NOTE:**

For some background on this topic, refer to [“Polygon Smoothing/Simplifying” on page 4-31](#).

► **To change the Polygon Smoothing Threshold value:**

1. Click on Tools:Options:Polygon Threshold. The Polygon Smoothing Threshold dialog box pops up.
2. Enter a Polygon Smoothing/Simplifying Threshold value.



3. Click OK to exit.

**NOTE:**

For some background on this topic, refer to [“Polygon Smoothing/Simplifying” on page 4-31](#).

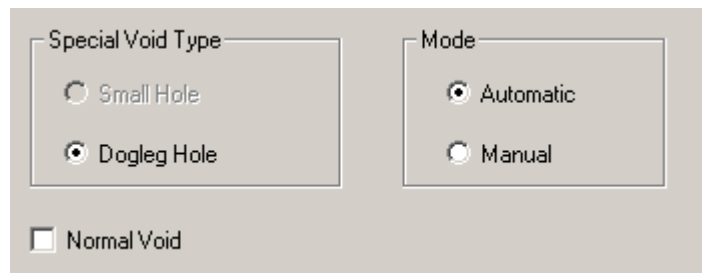
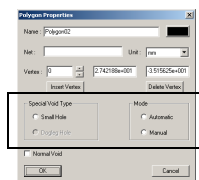
### Special Voids — Dog-Leg Holes and Small Holes

Whenever a trace has a change in any of its computational “reference planes” we detect and model it accordingly with *trace break* and *trace over split plane* functions — however, if a void is identified as a *special void* that void will receive special handling during simulation. We currently recognize two types of voids which can be identified on the basis of *size* and *content* criteria: *small holes* and *dog-leg holes*. The size criteria is established with the Special Voids Criteria dialog box — in addition, a *small hole* must not contain any other objects and a *dog-leg hole* must contain at least one trace and at least one via or one circuit connection to the trace.

- A “small hole” will be ignored during simulation.
- A “dog-leg hole” will be ignored if all vias passing through the hole or any circuit nodes inside the hole are outside the field computation domains with the presence of the hole.

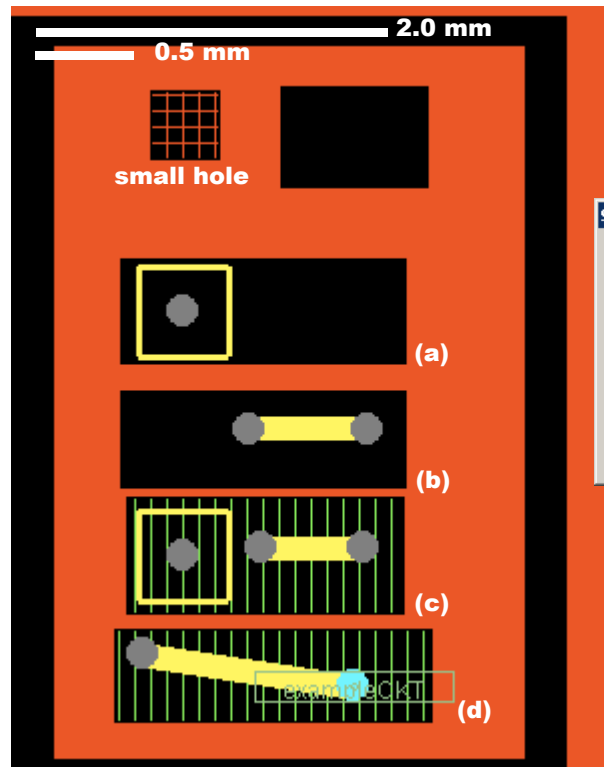
You may identify (or earmark) “special voids” in all package layers. For operational details refer to [“To establish special void size criteria:” on page 4-64](#). After establishing the size criteria, identification of special voids will proceed on an automatic basis, except the ability to manually identify voids also exists, refer to [“To manually select a void object as a “special void”:” on page 4-65](#). Voids that otherwise would be designated as “*special voids*”, on the basis of size and content criteria, can also be manually un-designated to be treated as “normal” voids, refer to [“To manually select a void object as a “normal void”:” on page 4-67](#). The current status of voids in the package can be determined visually (in SPDGEN) and from inspecting their properties dialog box. Press the *Shape Process* button to view the new (or latest) status of a void. Small voids are designated with a red mesh and dog-leg holes are designated with green vertical lines.

To summarize, in addition to other attributes for a shape component, several different types of void status are possible:

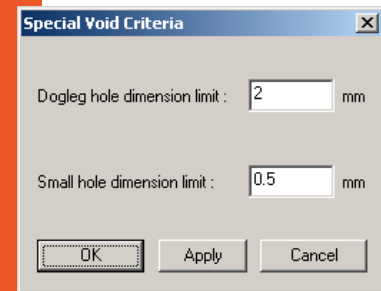


- Small Hole / Automatic — This void will be processed “automatically” on the basis of size and content criteria.

- Small Hole /Manual — This status designates the void as Special Void/Small Hole — even if it does not meet the size criteria.
- Dogleg Hole /Automatic — This void will be processed “automatically” on the basis of size criteria.
- Dogleg Hole /Manual — This status designates the void as Special Void/Dog-Leg Hole — even if it does not meet the size criteria.
- Normal Void — The void will be regarded as a normal void even if it does meet size and content criteria for Special Voids.



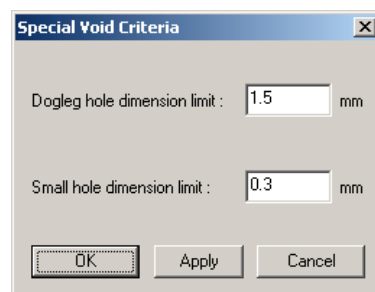
**For example:  
where Special Void Size  
criteria is:  
dog-leg hole = 2.0 mm  
small hole = .5 mm**



**a - size OK for dog-leg,  
but does not fulfill criteria  
b - size OK for dog-leg,  
but does not fulfill criteria  
c - size OK for dog-leg,  
and does fulfill criteria  
YES (dog-leg hole)  
d - size OK for dog-leg,  
and does fulfill criteria  
YES (dog-leg hole)**

► **To establish special void size criteria:**

1. With the Layer View window active, select Tools:Options:Special Void Criteria, the Special Void Criteria dialog box appears.
2. Enter the length any axis of the hole must be no larger than.



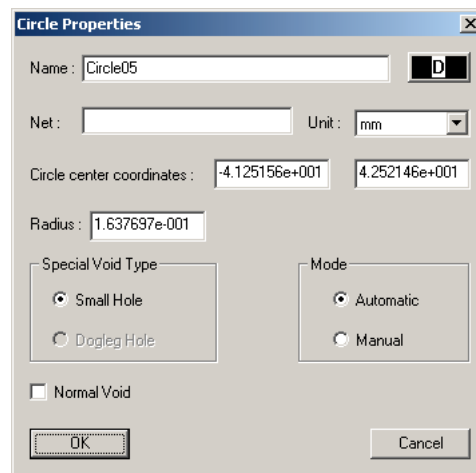
3. Click OK to enter the new value and dismiss the dialog box.

**NOTE:**

The default value for dog-leg hole is 1.5 mm — that is, no axis (in the x or y direction) of the hole can be longer than 1.5 mm. The default value for small hole is .3 mm — that is, no axis (in the x or y direction) of the hole can be longer than .3 mm. **Special voids should not be arbitrarily large because they introduce approximations in simulations.**

► **To manually select a void object as a “special void”:**

1. Bring out the property menu of the shape component of interest by right-clicking on the shape of interest, the shape’s Properties dialog box appears. Refer to [“To view a shape component’s property box:” on page 4-49](#). Note the Special Void Type and Mode sections are active and they indicate the current “special void” status of the void.

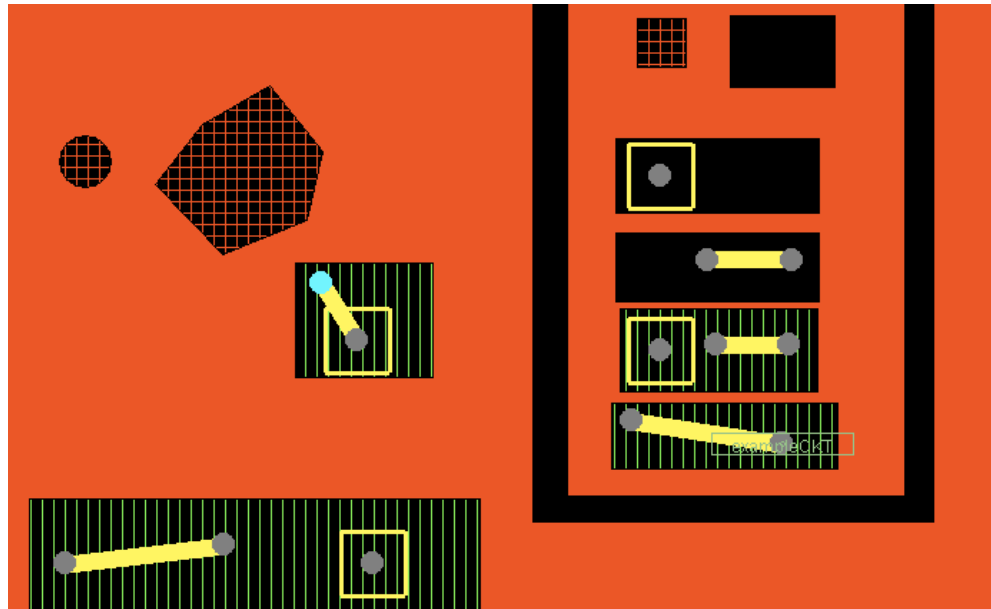


Shape Process button

**NOTE:**

If the Special Void Type and Mode areas are not active, the unionize shapes process must be run. Press the *Shape Process* button to clip traces and unionize shapes; or, save the file and select shape unionize function. Refer to [“Shape \(and trace\) Process function” on page 4-30](#), [“To unionize shapes \(only\):” on page 4-55](#), [“To segment and clip traces, unionize shapes, highlight special voids:” on page 4-61](#) and/or [“To clip traces and unionize shapes:” on page 4-59](#). After the shape unionize function has processed the file, the Special Void and Mode sections will be active.

2. Click on the Manual option, then click OK to enter the new status for the void and dismiss the dialog box. Now the void will be designated Special Void/Small Hole (or Special Void/Dog-Leg Hole) even if it does not meet the size criteria. Refer to [“Special Voids — Dog-Leg Holes and Small Holes”](#) on page 4-63 for more information.
3. Press the *Shape Process* button to view the new status of the void. Small voids are designated with a red mesh and dog-leg holes are designated with green vertical lines.

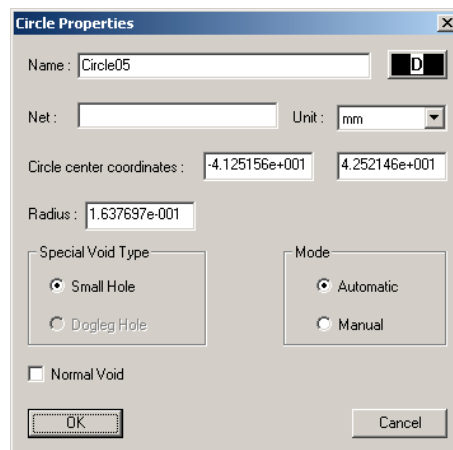




► **To manually select a void object as a “normal void”:**

Sometimes you want to override the automatic criteria set for special voids. Here’s how to do it.

1. Bring out the property menu of the shape component of interest by right-clicking on the shape of interest, the shape’s Properties dialog box appears. Refer to [“To view a shape component’s property box:” on page 4-49](#). Note the Special Void Type and Mode sections are active and they indicate the current “special void” status of the void.



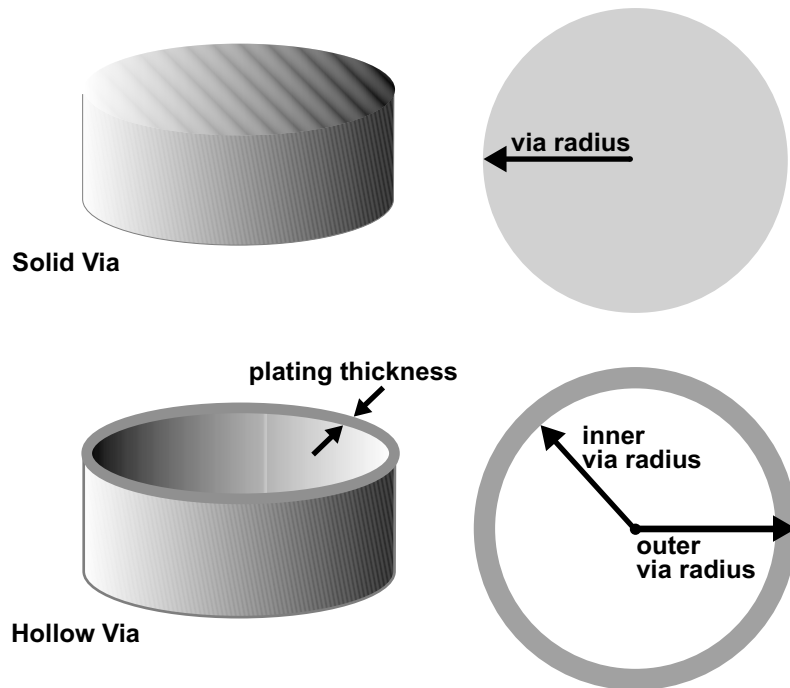
**NOTE:**

If the Special Void Type and Mode areas are not active, the unionize shapes process must be run. Press the *Shape Process* button to clip traces and unionize shapes; or, save the file and select shape unionize function. Refer to [“Shape \(and trace\) Process function” on page 4-30](#), [“To segment and clip traces, unionize shapes, highlight special voids:” on page 4-61](#) and [“To clip traces and unionize shapes:” on page 4-59](#). After the shape unionize function has processed the file, the Special Void and Mode sections will be active.

2. Click on the Normal Void option, then click OK to enter the new status for the void and dismiss the dialog box. Now the void will be designated “normal” even if it does meet the size and content criteria for a special void. Refer to [“Special Voids — Dog-Leg Holes and Small Holes” on page 4-63](#) for more information.

### Working on Vias

Vias are vertical interconnections between different metal layers. A via can be solid or a hollow type via with a “plating thickness”.



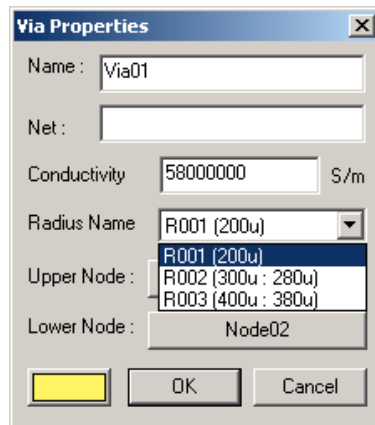
Vias can be added to the .spd file directly or they may be present as a result of translation of a file from another program. Properties tracked in the .spd file for vias include: name, net, conductivity, location (package node and layer), plating thickness, inner via radius, outer via radius, via type (solid, hollow),

Managing and assigning properties to vias is a two stage process: first, a suitable *via radius name definition*, which defines via geometry and thickness must exist — then, the via must be associated with the appropriate definition as well as the other properties. The Via Radius Map dialog box is used to create and manage *via radius name definitions*. The Via Properties dialog box is used to establish other properties of the package vias.

### Via Radius Map

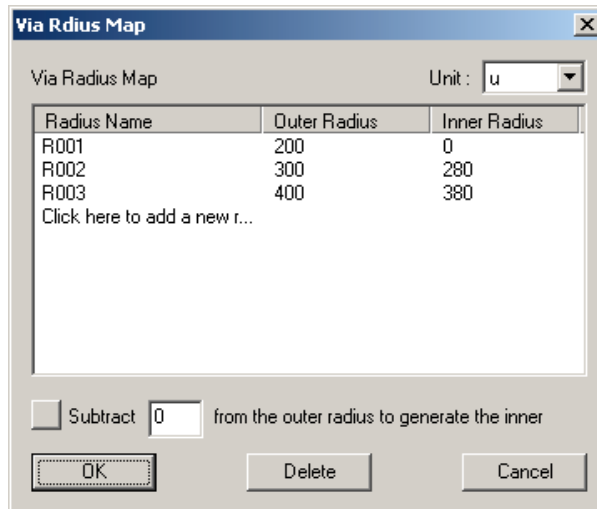
Radius Name definitions enable you to assign via geometry and thickness properties to individual vias using the Via Properties dialog box. As shown below, the outer and inner radius values are separated by a colon (outer radius value:inner radius value) in the parenthesis following each Radius Name available in the Via Properties dialog box. These “Radius Names” and their associated radius values are established using the Via Radius Map dialog box

discussed in this section. For more information on the Via Properties dialog box refer to [“To add a new via:” on page 4-74.](#)



These values are established with the Via Radius Map dialog box.

Click Edit:Via:Via Radius, to view the Via Radius Map dialog box.



The Via Radius Map dialog box enables you to:

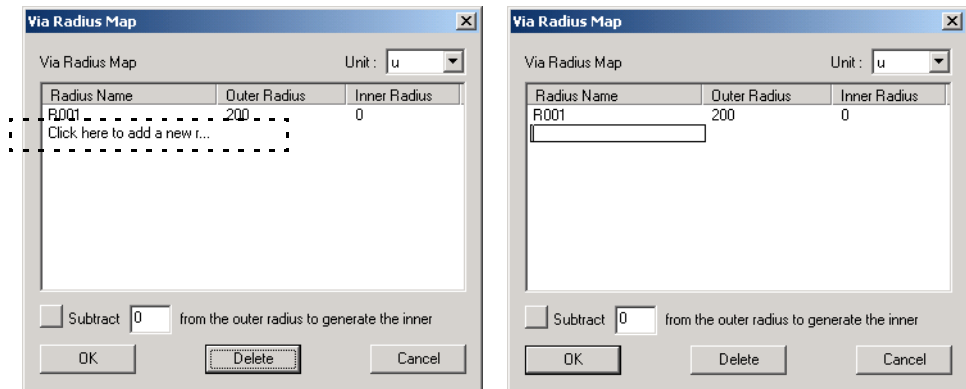
- view via radius name definitions,
- add new via radius definitions,
- edit via radius definitions
- and, delete via radius name definitions.

Let us discuss the Via Radius Map dialog box shown above: We see that ‘R001’ is a **Radius Name**. This name is associated (internally) with a value derived by subtracting the **Inner Radius** value from the **Outer Radius** value. As shown, R001 has an outer via radius of 200 microns and an inner radius value of zero; thus, this via radius name definition is used *to represent solid vias* of 400 micron

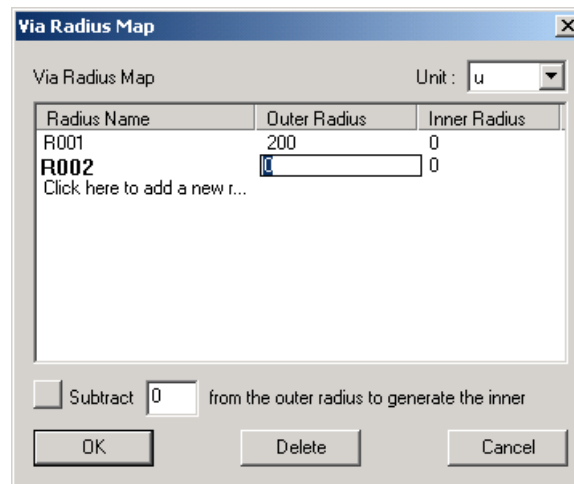
diameter. The **Outer Radius** value for R002 equals 300 microns and the **Inner Radius** value equals 280 microns — thus, R002 is used *to represent hollow vias* of 600 micron diameter and 20 micron plating thickness. The Via Radius Map dialog box also enables the user to globally specify a via plating thickness — refer to the section, [“To globally specify the via plating thickness:” on page 4-72](#) for more information. The units of via radius can be specified in microns(u), millimeters(mm) or mils(mil).

► **To add a new via radius name definition:**

1. Click Edit: Via:Via Radius Map (from the Layer View window menu bar). The Via Radius Map dialog box appears.
2. In the Via Radius Map dialog box, click the dialog box area titled “Click here to add a new radius”. A field entry area appears.



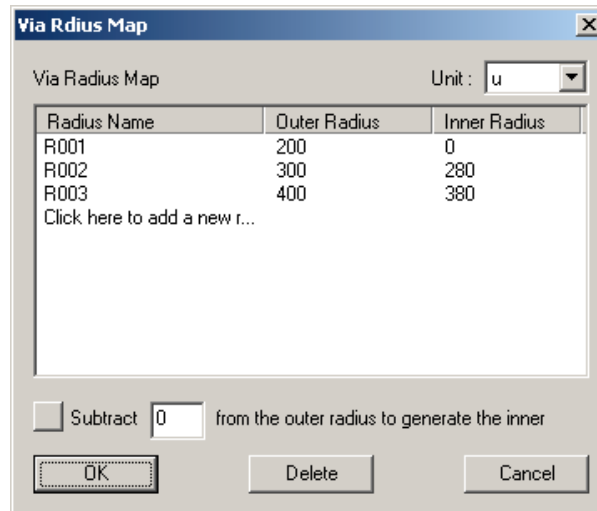
3. Enter the name of the new via radius definition and press Enter.



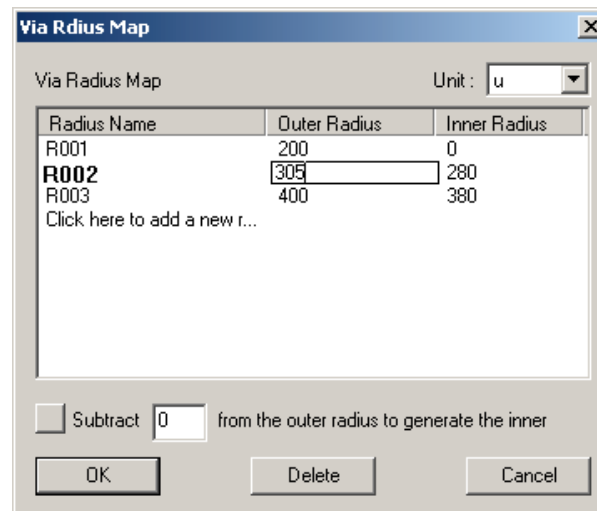
4. Observe the Outer and Inner Radius default values displayed — for a new name they will be zero. To change the values, left-click on a value. The value field becomes active and may be edited.
5. Press the OK button to exit the dialog box and save the information.

► **To change a via radius name definition:**

1. Click Edit: Via: Via Radius Map. The Via Radius Map dialog box appears.



2. Observe the Inner or Outer Radius values. Left-click the value you wish to change — it will become active and you can edit the value.

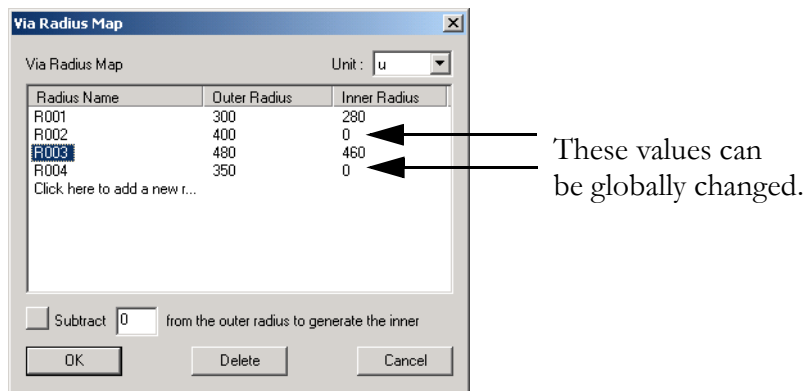


3. Click the OK button to exit the Via Radius Map dialog box and save the new information entered.

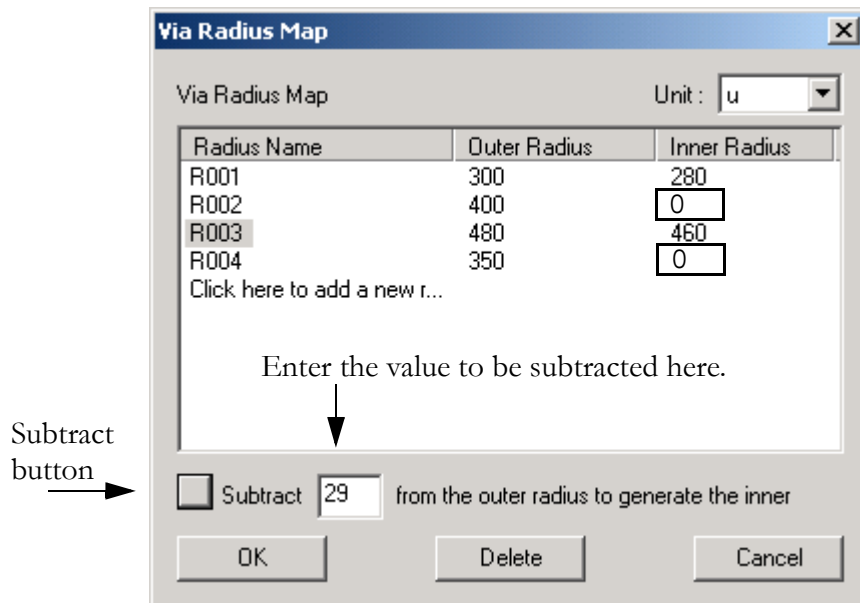
► **To globally specify the via plating thickness:**

The via plating thickness is typically derived as the difference between the outer radius and the inner radius values. Many CAD tools do not record an inner via radius value; thus, in a .spd file obtained from translation, the Radius Name definition may not have an Inner Radius value specified. This procedure enables you to conveniently specify a plating thickness for all Via Radius Name definitions that have an inner radius value of zero.

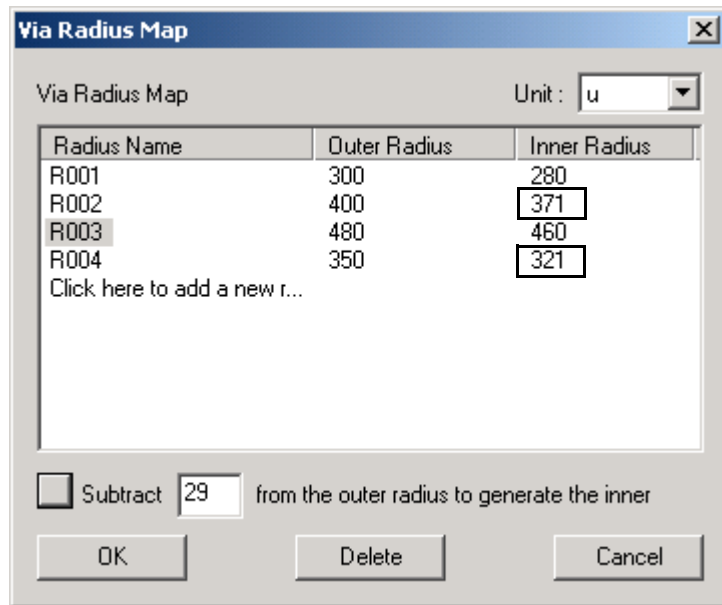
1. Click Edit:Via:Via Radius Map (from the Layer View window menu bar). The Via Radius Map dialog box appears. Observe the Inner and Outer Radius values. The Radius names with Inner Radius value equal to zero can be globally changed.



2. Enter a value representing the plating thickness to the blank field, "Subtract (blank) from the outer radius to generate the inner radius". **The value entered cannot be larger than the Outer Radius value specified.**



3. Press the *Subtract* button. Observe a new Inner Radius value has been generated based on the value entered.



4. Click OK to exit and save the new information — or, click Cancel to exit without saving the generated values.

► **To delete a via radius name definition:**

**NOTE:**

You cannot delete a via radius name definition that is in use.

1. Click Edit:Via:Via Radius Map. The Via Radius Map dialog box appears.
2. Select the radius name from the list box (it will highlight) then click the Delete button (or, press the Delete key). **The via radius definition will be deleted if no via in the package uses it.**
3. Click the OK button to exit the Via Radius Map dialog box and save the new information entered.

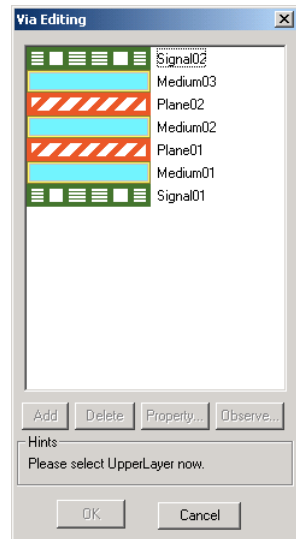
**NOTE:**

To conveniently “change” a via radius name definition in use by many vias you may edit the .spd (text-based) file using a text editor’s find and replace option.

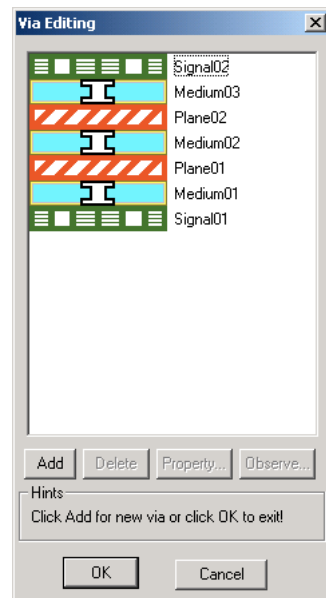


► **To add a new via:**

1. Make sure the Layer View window is active. Click the *Add Via* button on the tool bar. The cursor appears as a **+** as it moves into the Layer View window.
2. Move the cursor to where you want the via to be located and click the left mouse button once. The Via Editing dialog box appears.



3. In the Via Editing dialog box, click the left mouse button on the upper metal layer and then on the lower metal layer that the via connects to.
4. If more sections are needed, click the *Add* button in the Via Editing dialog box, and click the upper and lower metal layers for each via section.
5. Repeat the last step until all vertical connections are specified.



6. Click the *OK* button to exit the Via Editing dialog box. The new via will appear in the Layer View Window.



**Via Representation On the Layer View Window**

On the Layer View window (which depicts a top-down view of the package) vias are represented in conjunction with the following symbols:


- ☐ Represents the upper end of a via
- ✕ Represents the lower end of a via
- ⊙ Represents a via passing through the plane



Add Via

**► To add a via at an existing trace node:**

If you want to add a via to connect an existing trace, you can use the snap to node function to add a via right at the trace node by the following steps:

1. Make sure the current active layer contains the trace. If not, activate the layer in the Layer Selection dialog box.
2. Click the *Add Via* button from the tool bar. The cursor appears as a **+** as it moves into the Layer View window.
3. Move the cursor close to the trace node where you want the via to be connected with and **hold down the Ctrl key**. The shape of the cursor should be transformed to . Then click the left mouse button. The Via Editing dialog box appears.
4. Add a via as described in the previous section [“To add a new via:” on page 4-74](#).



Move button

**► To move a via:**

1. Make sure the current active layer contains the via. If not, activate the layer in the Layer Selection dialog box.
2. Click the Move button on the tool bar. The cursor appears as a **+** as it moves into the Layer View window.
3. Click the left mouse button at the node to be selected. A cyan-color dot will appear at the location of the selected node.
4. Hold down the left mouse button and drag the selected node to the desired location.


**NOTE:**

You can move a via to a precise location by entering the coordinates of the node. See the section [“To move a node by entering the coordinates:” on page 4-26](#).



Move button

► **To move a via and snap it to a trace node:**

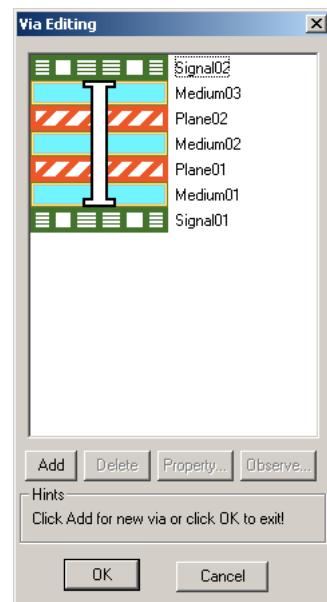
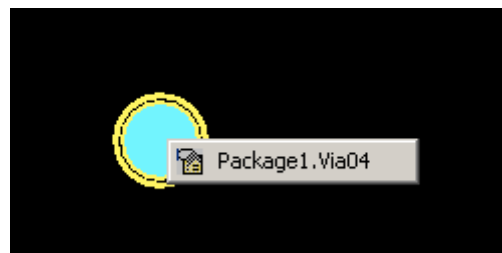
1. Make sure the current active layer contains the via and the trace of interest. If not, activate the layer in the Layer Selection dialog box.
2. Click the *Move* button on the tool bar. The cursor appears as a **+** as it moves into the Layer View window.
3. Click the left mouse button at the node to be selected. A cyan-color dot will appear at the location of the selected node.
4. Hold down the left mouse button and drag the selected node to the desired trace node while holding the Ctrl key (the shape of the cursor should be transformed to ) , then release the left mouse button.

► **To edit the properties of a via individually:**

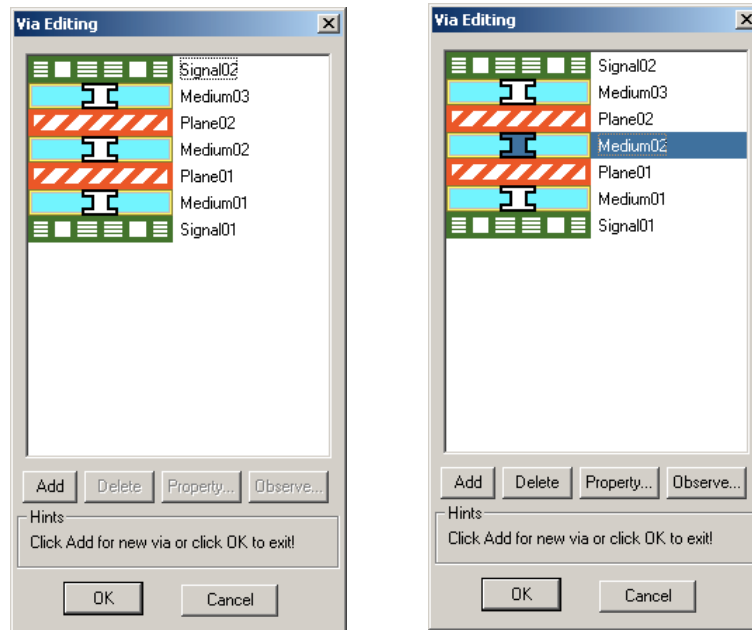
*Via properties may be edited individually or as a group.*

1. Make sure the current active layer contains the via. If not, activate the layer in the Layer Selection dialog box.
2. Invoke the Via Editing dialog box. Use the right mouse button to directly click on the via, then select the via name in the fly-out menu; or, use the Select button to select the via then click the property button, the Via Editing dialog box appears.

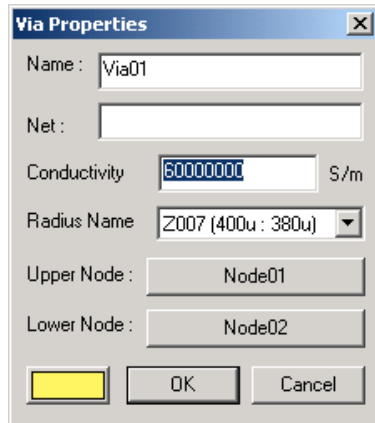
Select button



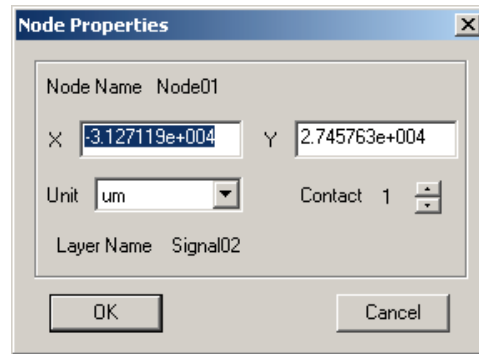
3. In the Via Editing dialog box, if multiple vias appear, click the left mouse button on the via to be edited. The selected via section will be highlighted.



4. Click the *Property* button in the Via Editing dialog box. The Via Properties dialog box, for the via selected, appears.



5. In the Via Properties dialog box, you can change the name of the via (which is initially generated by the program), the net the via is associated with, the conductivity of the via, the name of the via radius definition, the upper and lower node connection locations and the color of the via.
6. You can also inspect the nodes of the via. Click the *Upper Node* or the *Lower Node* buttons, the Node Properties dialog box appears. For details refer to [“Using Node Tools” on page 4-25](#).



7. Click the *OK* button to exit the Node Properties dialog box.
8. Click the *OK* button to exit the Via Properties dialog box.
9. Click the *OK* button to exit the Via Editing dialog box.

► **To edit the properties of several vias at once:**

*Several via properties may be edited individually or as a group: Net Name, Conductivity, Color and Radius Name.*

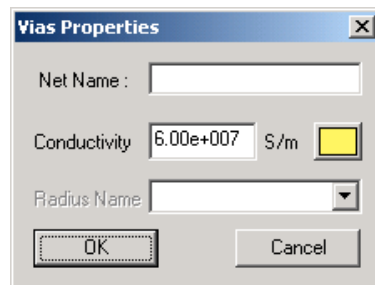


Select button



Property button

1. Make sure the current active layer contains the vias of interest. If not, activate the layer in the Layer Selection dialog box.
2. Use the *Select* button to select several vias at once. Click on the *Select* button, then drag across the vias of interest.
3. When the vias are highlighted, then click the *Properties* button. The Vias Properties dialog box appears.



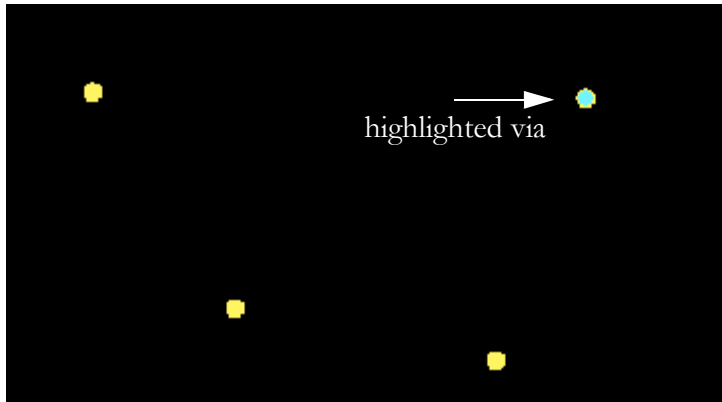
4. In the Vias Properties dialog box, you can change the net the vias are associated with, the conductivity of the vias selected, the name of the via radius definition (for the group of vias selected), and the color of the vias.
5. Click the *OK* button to exit the Via Properties dialog box.

► To delete a via (1 of 2):



Select button

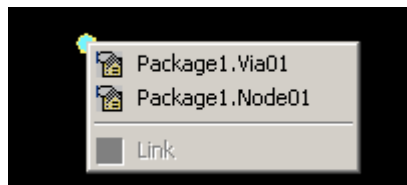
1. Select the via. Make sure the Layer View window is active and the via of interest is visible. Activate the *Select* button by clicking on it, then select the via. The via will be highlighted.



2. Press the Delete key on the keyboard to delete the selected via.

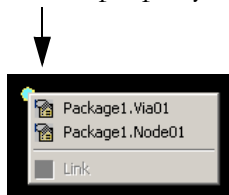
► To delete a via (2 of 2):

1. Select the via. Make sure the Layer View window is active and the via of interest is visible, then right-click on the via. A small flying menu appears with details of the via and the node.

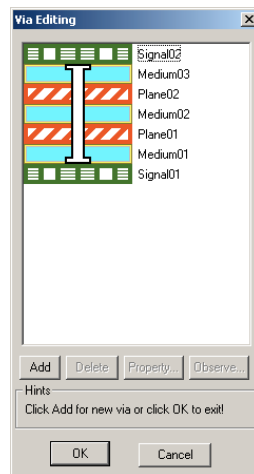


2. Click on the Property icon to bring out the Via Editing dialog box.

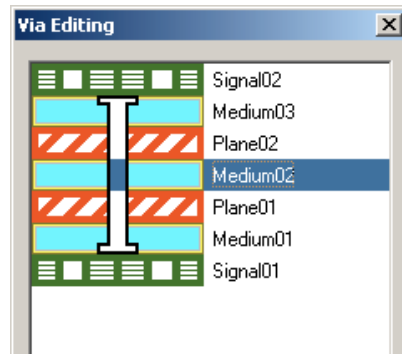
Click the property icon.



The Via Editing dialog box will appear.



3. In the Via Editing dialog box, click the left mouse button on the via to be deleted.



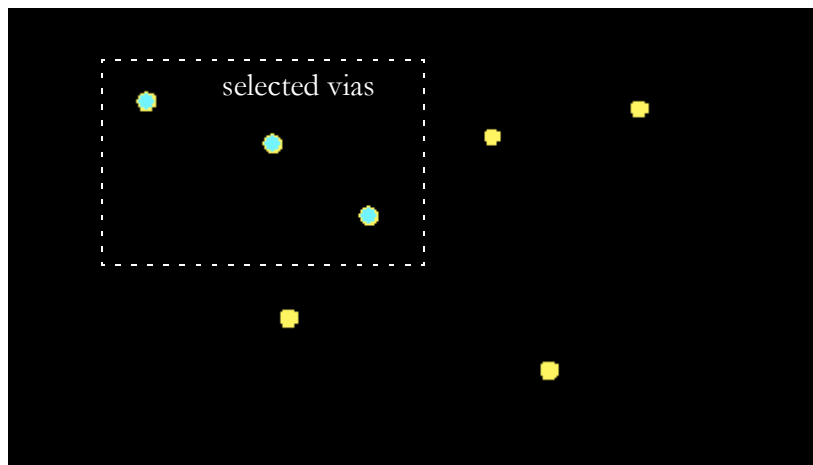
4. Click the *Delete* button in the Via Editing dialog box.
5. Click the *OK* button to exit the Via Editing dialog box.

► **To copy vias (using the Copy Via[s] To button):**


**NOTE:**

The vias that are generated through copying will connect to the same layers and have the same properties as the original vias.

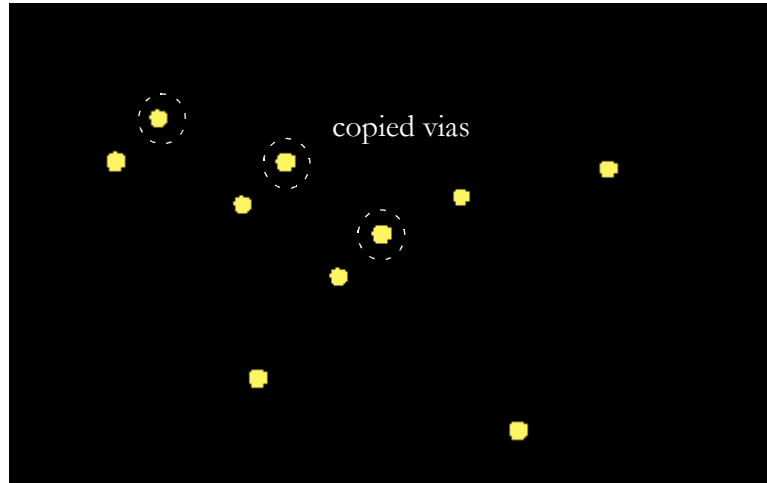
1. Select one or more vias. Make sure the Layer View window is active and the vias of interest are visible. Activate the *Select* button by clicking on it, then select the vias of interest by holding down the left mouse key and dragging. The vias selected will be highlighted.



Copy Via(s) To

2. Click the *Copy Via(s) To* button on the tool bar. The cursor appears as .

3. Move the cursor to where you want the vias to be located and click the left mouse button — the vias will be added. You can continue to click a number of locations, and at each location you click, the vias are added there.

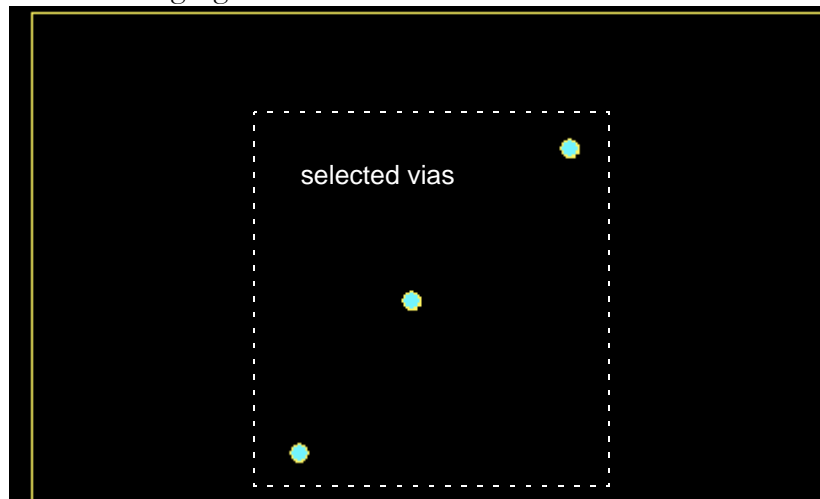


► To copy vias (using the Shift Copy Vias menu selection):

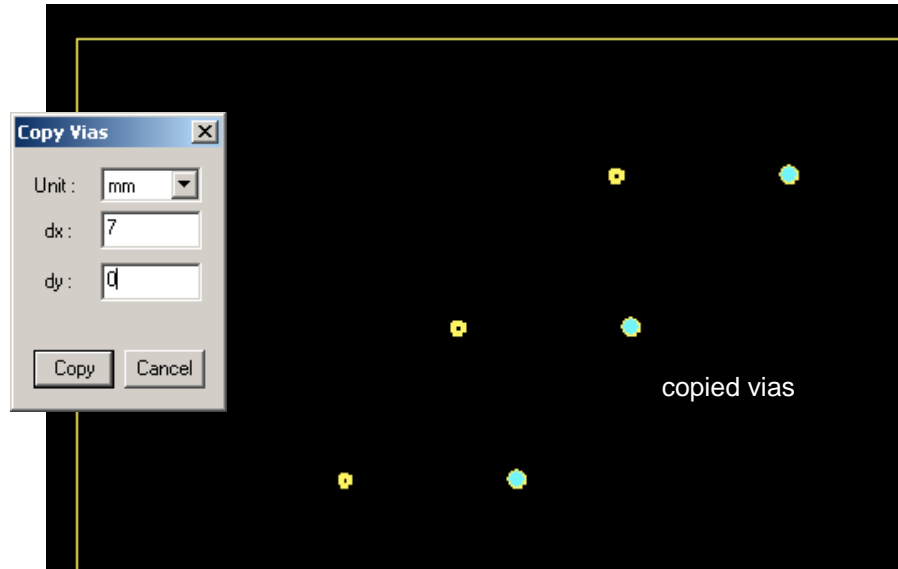
**NOTE:**

The vias that are generated through copying will connect to the same layers and have the same properties as the original vias.

1. Select one or more vias. Make sure the Layer View window is active and the vias of interest are visible. Activate the *Select* button by clicking on it, then select the vias of interest by holding down the left mouse key and dragging. The vias selected will be highlighted.

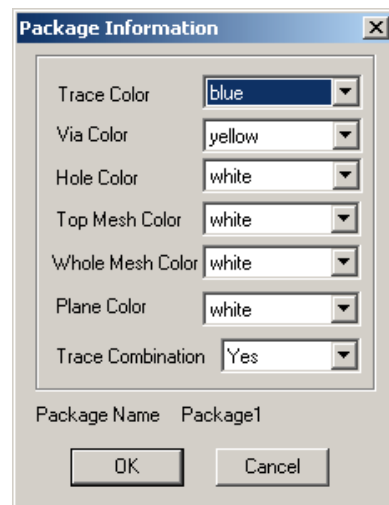


2. Select Tools:Shift Copy Vias after the via/vias of interest are selected.
3. In the Copy Vias dialog box, enter dx and dy. This specifies where the new vias will be placed, with respect to the original vias, in the x and the y dimensions.



► **To set the color for via display:**

1. Click Tools: Options: Component Color (from the Layer View window menu bar). The Package Information dialog box appears.
2. Select the default color for the display of vias.
3. Click the OK button to exit the Package Information dialog box.

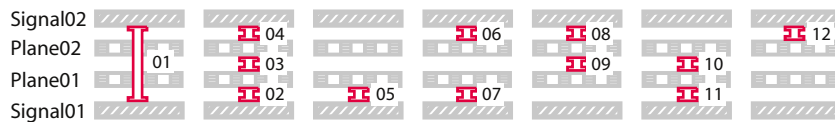




► **To insert vias using existing vias:**

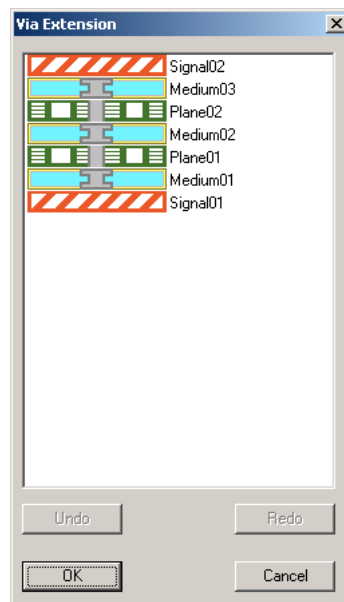
Vias provide vertical interconnections between layers in the package; sometimes, you want to conveniently insert vertical interconnections (vias) in conjunction with existing vias. For example, if you have merged two packages they are initially separated after the merge (by air) and must be connected. If you want to make vertical connections using vias and nodes, please refer to [“To insert vias using existing package nodes and vias:” on page 4-85](#).

1. Begin by selecting the via (or vias) of interest. If necessary, refer to [“Selecting Objects” on page 4-21](#). As an example, the figure below represents, in cross-section, the vias that we have selected to extend. As can be seen, Via01 is terminated at Signal02 and Signal01, via04 is terminated at Signal02 and Plane02, via03 is terminated at Signal02 and Plane01, and so on...

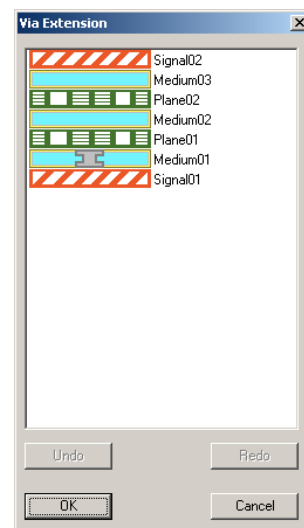


Extend Vias

2. Click the *Extend Vias* button on the tool bar, the Via Extension dialog box appears. The grayed via indicators show what layers the selected vias are located on. Note, in this figure (part a) the indicators overlap because some vias span layers and there is more than one via on a layer; however, had we selected only one via, say via05, it would appear as shown in part b.



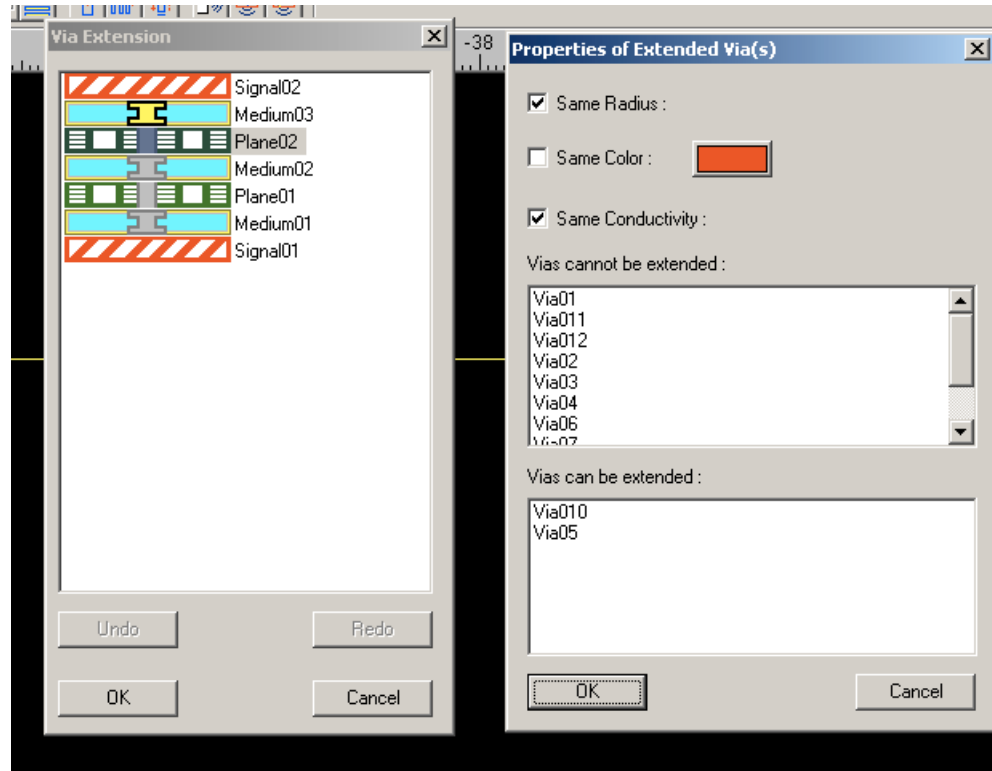
-a-



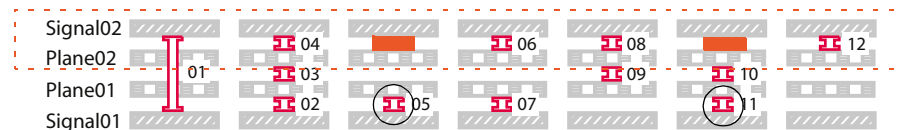
-b-

3. Click the layers to select the layer pair you want the new extended via (or vias) to connect. A yellow via appears on the Via Extension dialog box to reflect your

completed selection and the Properties of Extended Via(s) dialog box appears. This dialog box is designed to report the properties of the subset of the selected vias *that can be extended*. Oftentimes, not all vias selected can be extended.

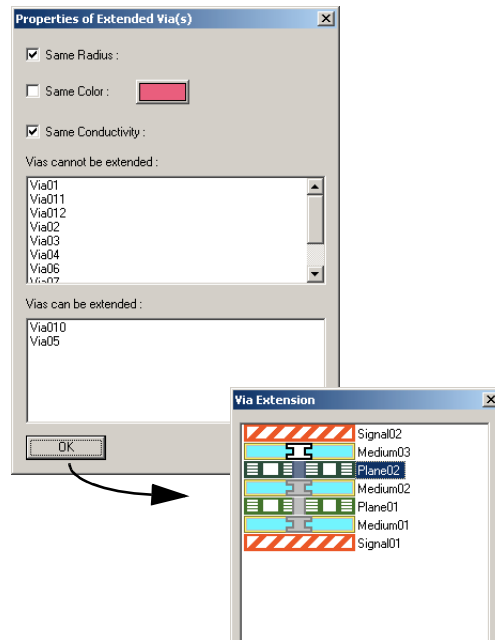


4. Let us review the vias we selected (for the extending vias procedure) and see which vias could not be extended to connect layers Signal02 and Plane02, as indicated in the Properties Of Extended Via(s) dialog box. Extensions cannot occur which would interfere with existing vias — so that is not permitted. The dotted box below indicates the layer pair selected, Signal02 and Plane02. Vias that are “extendable” to the layer pair are circled and red boxes indicate where the new vias will occur.



5. Note the circled vias above. New vias can inherit the properties of vias from which they stem (Same Radius, Same Color, Same Conductivity) or uncheck the checkmark box that precedes each property to assign a new radius, color and conductivity properties for the new vias. Click the OK box when you have made

your selections. A white via will appear in the Via Extension dialog box to reflect the system has recorded your selection.



6. If desired, select another layer pair to extend the selected vias to. When you are done, click OK to enter your via extension choices and dismiss the Via Extension dialog box. What you will see on the screen at the completion of the process depends upon the layer that is active. As you inspect the layers recall, a square symbol represents the upper end of a via, an 'X' represents the lower end of the via and two circles represent a via passing through the plane. If necessary, click View:Layer Selection... to activate the layers that contain the end nodes or clearance holes for the vias of interest.

- ☐ Represents the upper end of a via
- ☒ Represents the lower end of a via
- ☒ Represents a via passing through the plane

► **To insert vias using existing package nodes and vias:**

Vias provide vertical interconnections between layers in the package; sometimes, you want to conveniently insert vertical interconnections (vias) in conjunction with existing vias and package nodes. For example, if you have merged two packages they are initially separated after the merge (by air) and must be

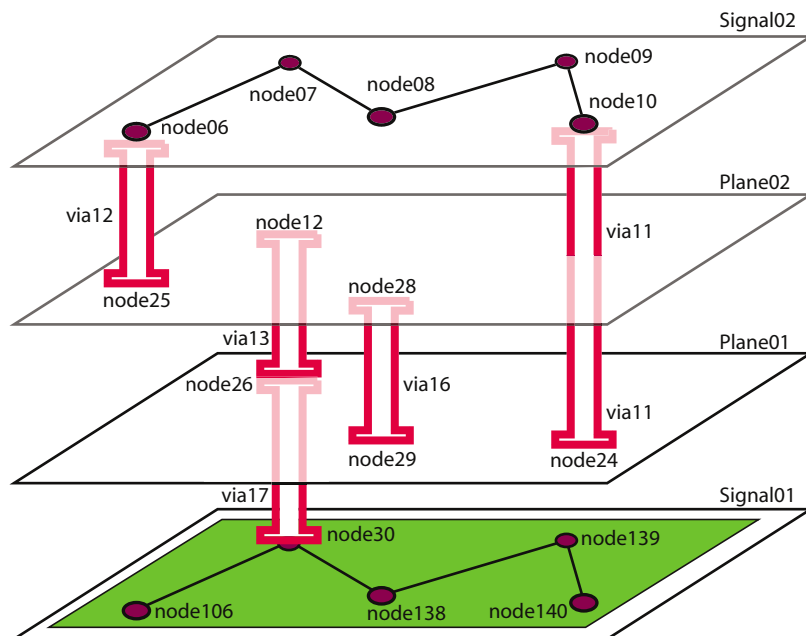
connected. If you only want to extend vias, please refer to [“To insert vias using existing vias:” on page 4-83](#).

1. Begin by selecting the package nodes and vias of interest. If necessary, refer to [“Selecting Objects” on page 4-21](#).

**NOTE:**

When selecting objects on several layers, remember to UNclick the *View Only Active Layer* option of the Layer Selection dialog box.

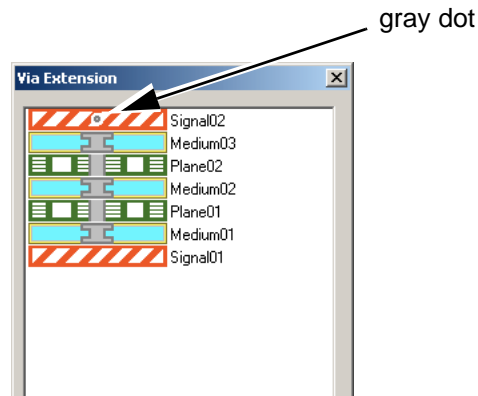
As a demonstration, the figure below represents the nodes and vias that we have selected to extend vertically — they are: node06, node07, node08, node09 and node10, via12, via17, via13, via16 and via11. As can be seen, Via 12 is terminated at node06 on layer Signal02 and node25 on layer Plane02; node07 does not extend vertically through a via — but, via13 is in vertical proximity (to node07) as is via17 — note via17 and via13 share node26; node28 is the upper node of via16 — node29 is the lower node which is in vertical proximity to node138 of layer Signal01, and so on... Via11 is included to complement the demonstration. The function to insert vias performs in the following manner: Isolated node objects will be extended vertically, by the insertion of vias — where a via already exists, the via will be extended.





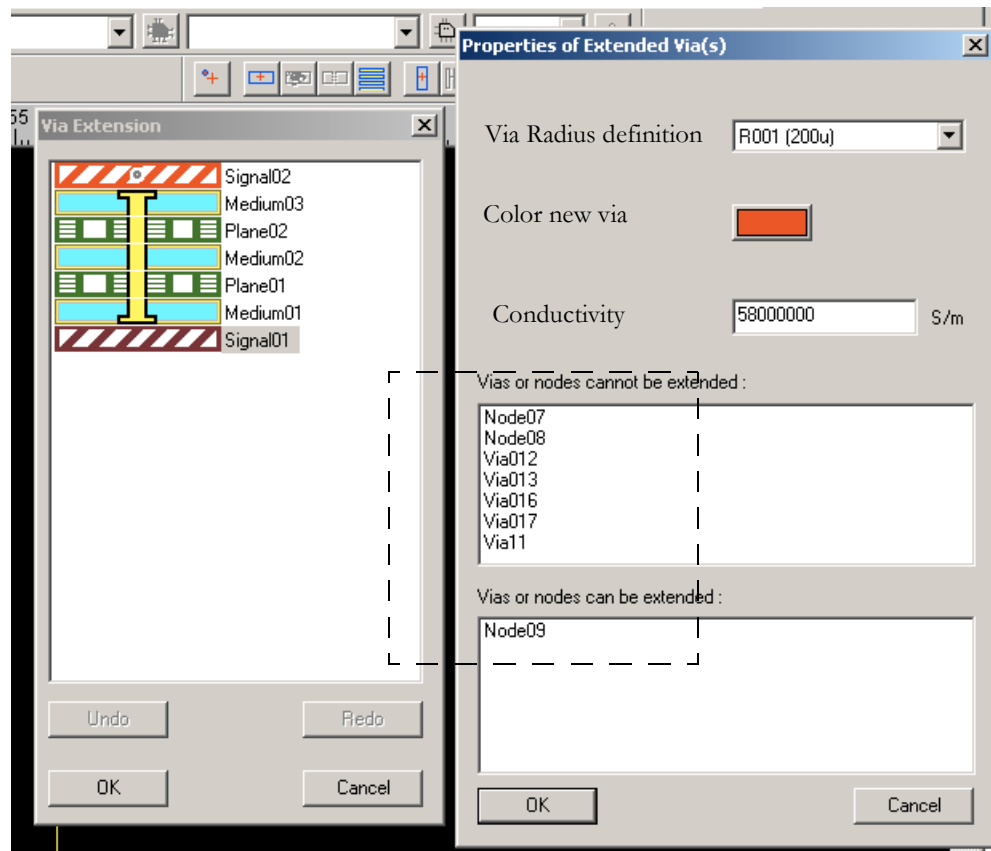
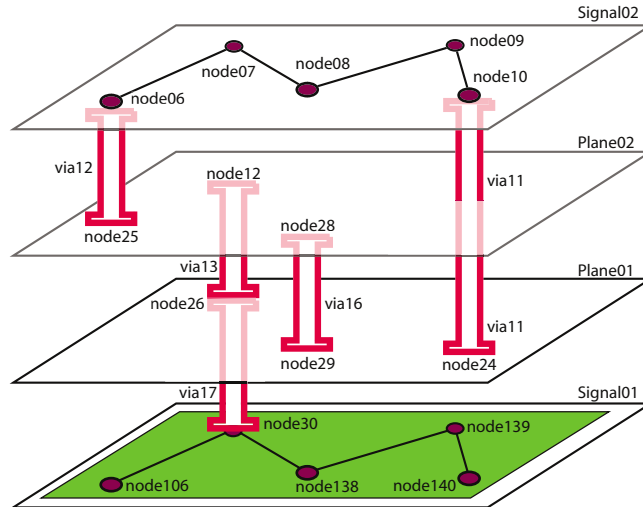
Extend Vias

2. Click the *Extend Vias* button on the tool bar, the Via Extension dialog box appears. The gray dot indicator shows the layer that contains the selected nodes. The grayed vias indicate the layers that contain vias.

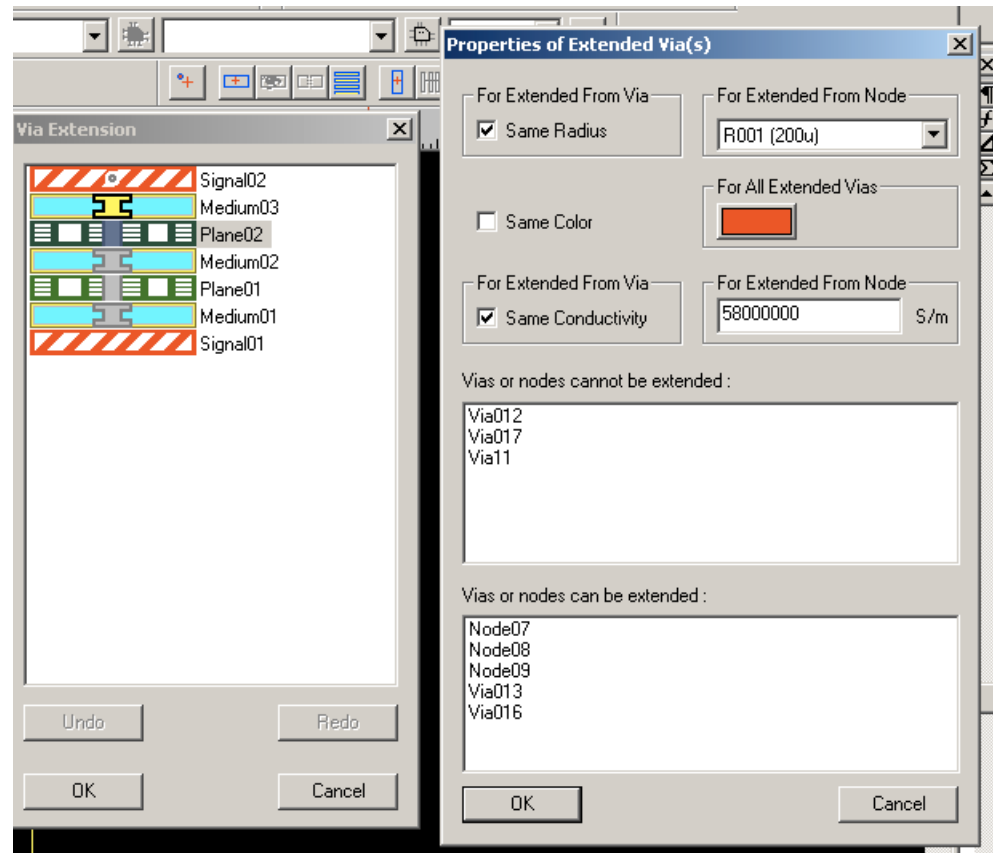
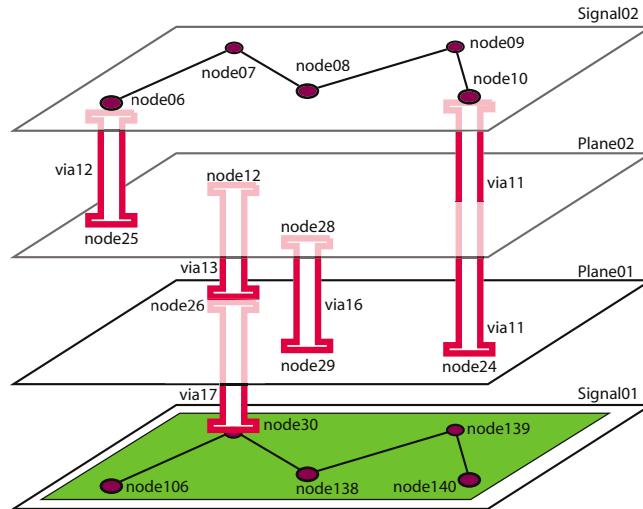


3. Click the layers to select the layer pair you want the newly *inserted* vias and/or *extended* vias to connect. A yellow via appears on the Via Extension dialog box to reflect your completed selection and a dialog box, Properties of Extended Via(s), appears to report the properties of the subset of the selected vias and nodes that can be extended. Oftentimes, not all vias and nodes selected can be extended. Let us look at some demonstration cases to see the results.

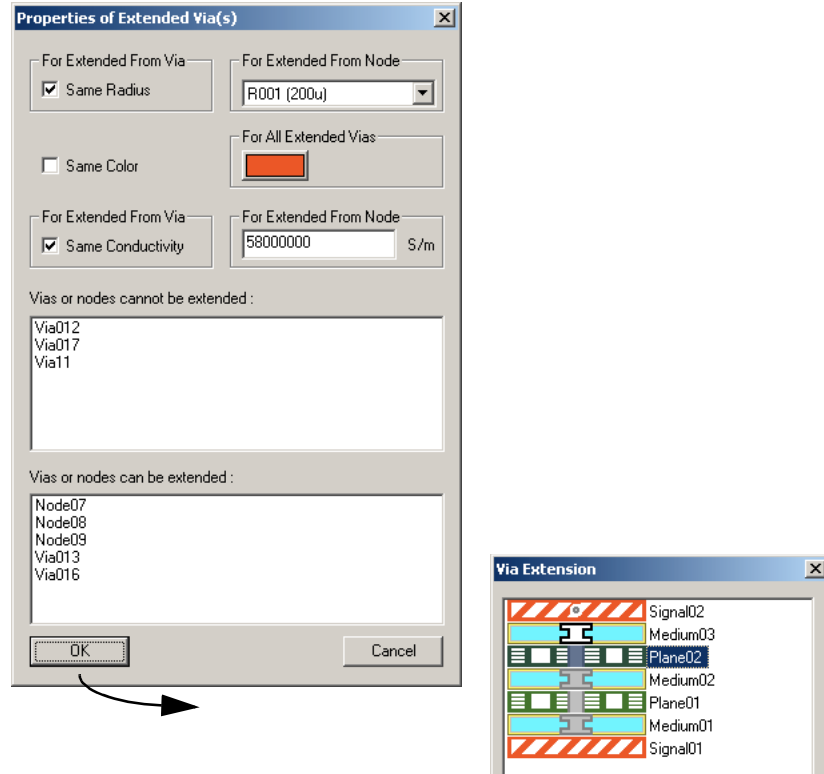
CASE 1 — The yellow via shows we want to extend nodes selected from layer Signal02 to layer Signal01. Based upon the example node/via configuration presented (in step 1) we see that only Node09 can be extended as indicated. All other nodes have existing vias blocking the extension.



CASE 2 — The yellow via shows we want to extend all nodes/vias from layer Signal02 to layer Plane02. Based upon the example node/via configuration presented (in step 1) we see that nodes 9, 8, and 7; and vias 13 and 16 can be extended as indicated. Vias 12, 17 and 11 cannot be extended. Again, existing vias are blocking the nodes 6 and 10. Via17 is blocked by via13 — while via13 can be extended.



4. In addition, the new vias can inherit the properties of the vias from which they stem (Same Radius, Same Color, Same Conductivity) or uncheck the checkmark box that precedes each property to assign a new radius, color and conductivity properties for the new vias. Click the OK box when you have made your selections. A white via will appear in the Via Extension dialog box to reflect the system has recorded your selection.



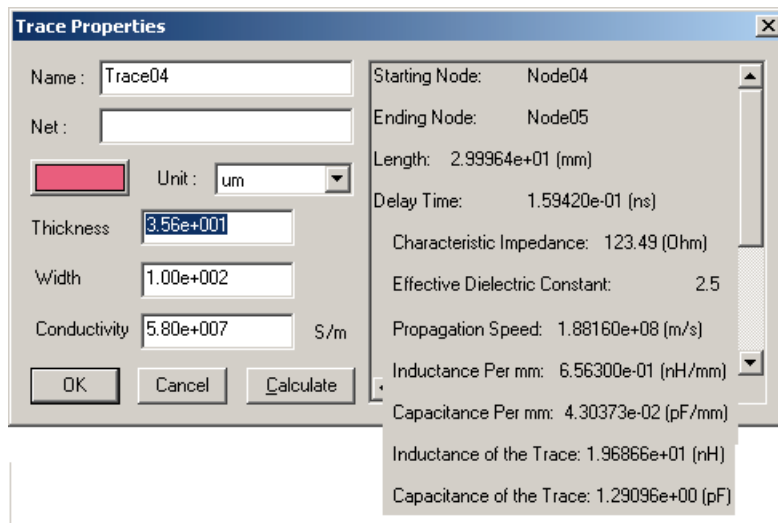
5. Select another layer pair to extend the selected nodes and vias to if desired. When you are done, click OK to insert the new via and dismiss the Via Extension dialog box. What you will see on the screen at the completion of the process depends upon the layer that is active. As you inspect the layers, a square symbol represents the upper end of a via, an 'X' represents the lower end of the via and two circles represents a via passing through the plane. If necessary, click View:Layer Selection... to activate the layers that contain the end nodes or clearance holes for the vias of interest. You may also view the new structure in 3D.

- ☐ Represents the upper end of a via
- ☒ Represents the lower end of a via
- ☒ Represents a via passing through the plane



### Working on Traces

Traces represent routed electrical connections. Traces can only be placed on signal layers — therefore, always activate a signal layer to work with trace tools. For every trace in your package SPEED2000 reports the: starting node, ending node, length, delay time, characteristic impedance, effective dielectric constant, propagation speed, inductance per mm, capacitance per mm, inductance of the trace and capacitance of the trace. Editable trace properties include: name, net, color, unit (for calculations), thickness, width and conductivity. Shown in the figure is the trace properties dialog box. After editing properties you may press the *calculate* button to update the values reported in the dialog box.



#### NOTE:

Electrical properties of a trace displayed in the Trace Properties dialog box are computed by a Method of Moment based electromagnetic field solver. In computation, metal skin effect loss, dielectric loss and dispersion, may also be taken into account.

#### ► To add a trace:

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click the *Add Trace* button on the tool bar. The cursor changes to **+** as it moves into the Layer View window.
3. Move the cursor to where you want the trace to start and click the left mouse button once.
4. Move the cursor to the next node of the trace and click the left mouse button once.



Add trace

5. Repeat the last step until you finish the last node of the trace, then **double click the left mouse button to complete the trace drawing**.



**NOTE:**

After drawing a trace, you are still in the “adding trace mode”. **You can keep adding traces until you click the right mouse button to quit the “adding trace mode”.**



Add trace

► **To add a trace connecting existing package nodes:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click the Add Trace button on the tool bar. The cursor changes to  as it moves into the Layer View window.
3. Move the cursor close to the node where you want the trace to start and holding down the Ctrl key. When the cursor transforms to a , click the left mouse button once. The starting node of the trace is then snapped to the existing node.
4. Move the cursor to the next node of the trace and click the left mouse button once.
5. Repeat the last step and always remember to hold down the Ctrl key before you click the left mouse button if you want your trace node be snapped onto an existing node.
6. After adding all the nodes of the trace, **double click the left mouse button to complete the trace drawing**.

**NOTE:**

When you draw a trace connecting with some existing nodes of traces or vias, use the Ctrl key to snap the trace node to existing nodes.

► **To change properties of a trace:**

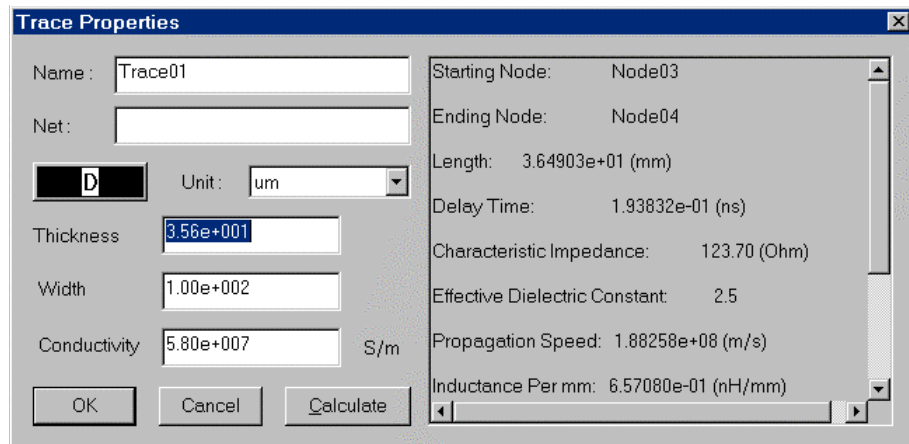
1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Select the trace. Click the *select* button. The cursor transforms to an outline arrow as it moves into the Layer View window. Click on the trace of interest. It will highlight when selected.
3. Click the *Property* button on the tool bar. The Trace Properties dialog box appears.



Select button



Property button



The **Trace Properties** dialog box contains the following fields and controls:

- Name:** Trace01
- Net:** (empty)
- Color:** D (Default)
- Unit:** um
- Thickness:** 3.56e+001
- Width:** 1.00e+002
- Conductivity:** 5.80e+007 S/m
- Buttons:** OK, Cancel, Calculate
- Starting Node:** Node03
- Ending Node:** Node04
- Length:** 3.64903e+01 (mm)
- Delay Time:** 1.93832e-01 (ns)
- Characteristic Impedance:** 123.70 (Ohm)
- Effective Dielectric Constant:** 2.5
- Propagation Speed:** 1.88258e+08 (m/s)
- Inductance Per mm:** 6.57080e-01 (nH/mm)

4. Change the properties of the trace in the Trace Properties dialog box. Press the *Calculate* button to view the characteristics of the modified trace.
5. Click the *OK* button to exit the Trace Properties dialog box.

**NOTE:**

If the color option in the Trace Properties dialog box is default (D), the trace will be drawn on screen using the color of the signal layer. However if you select another color for this trace, then the trace will have a distinctive color compared to other traces on the same signal layer. This provides a way to highlight a particular trace if you wish.



Select button

**► To delete a trace/traces:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Select a trace/traces using the Select button or the right mouse button.
3. Press the Delete key on the keyboard to delete the selected trace.

**► To move a node of a trace:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click on the Move button.
3. Select a node of the trace by clicking the left mouse button at the location of the node.
4. Hold down the left mouse button and drag the node to a new location.



Move Button

**► To move an entire trace:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click on the Move button.
3. Select on a middle point of a trace.
4. Hold down the left mouse button and drag the trace to a new location.



Move Button



Split Trace

► **To add a node on a trace:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Select a trace/traces using the Select button or the right mouse button.
3. Click the *Split Trace* button on the tool bar. A node will be added in the middle of the selected trace.




Move Button

► **To connect ends of two traces:**

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click on the *Move* button.
3. Select a node of a trace by clicking the left mouse button at the location of the node. Hold down the left mouse button and drag the cursor towards a node of another trace while holding down the Ctrl key (the cursor should be flashing), then release the left mouse button.

► **To connect a trace to a via:**

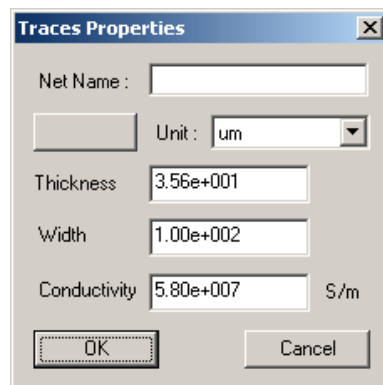
1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click on the *Move* button.
3. Select a node of the trace by clicking the left mouse button at the location of the node. Hold down the left mouse button and drag the cursor towards the node of the via while holding down the Ctrl key (the shape of the cursor should be transformed to ) , then release the left mouse button.

► **To change options for multiple traces:**



Select

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Select a group of traces.
3. Click on the Property dialog box to bring out the common properties of these traces. Enter new values as desired. Changes made in the dialog box will affect all the selected traces.



4. Click OK to exit the dialog box.

### Working on Coupled Lines

When traces are identified as coupled lines, the crosstalk between these lines will be calculated during the simulation. Coupled lines will be treated as multi-conductor transmission lines, whereas an isolated trace is modeled by the single transmission line algorithm. Coupled lines can be selected after relevant traces have been placed.



Couple Line Add

#### ► To identify coupled lines:

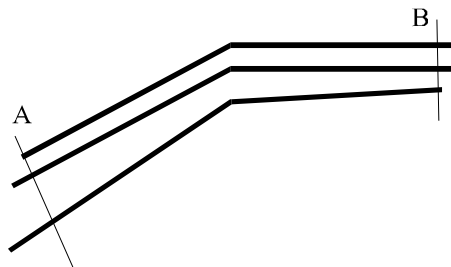
1. Click the *Couple Line Add* button on the tool bar, and move the cursor to the Layer View window.

#### NOTE:

If the *Couple line Add* button is not active, the file must be processed to: unionize shapes, clip and segment traces and highlight special voids. Refer to [“Pre-Conditioning the .spd File For Simulation” on page 4-55](#), [“To unionize shapes \(only\):” on page 4-55](#), [“To clip traces and unionize shapes:” on page 4-59](#), [“Traces Over “Split Planes”” on page 4-97](#) and [“To segment and clip traces, unionize shapes, highlight special voids:” on page 4-61](#).

**Tip:** Try to draw a line perpendicular to the average direction of the coupled traces and close to the starting nodes of the coupled traces.

2. Draw a straight line, called line A, across all the coupled traces as shown in the illustration diagram below. When you draw line A (while holding down the left mouse button), a yellow rubber-band will appear and move along with your mouse. After you release the mouse button, a yellow line will appear on the screen crossing all the coupled traces selected. Note that the direction of the yellow line may look different from the one you draw. This is due to the automatic adjustment program to ensure line A is perpendicular to the average direction of all the coupled traces. The purpose of drawing line A is to tell the program what are the coupled lines, and approximately where are the starting points of these coupled lines.



#### NOTE:

If you are not satisfied with the position of line A, click the right mouse button, and try again.

3. Draw second straight line, called line B, across all the coupled traces to mark the ends of the coupled lines. The ending nodes of these coupled lines will be automatically adjusted and marked. Note the ending nodes may not appear at the intercepting points of line B and the coupled traces. This is due to the automatic adjustment program to ensure all coupled traces have equal length.
4. A dialog box will appear to confirm the selection. If you think the highlighted coupled traces portion (in cyan color) is appropriate, click Yes, otherwise, click No and try again.

**NOTE:**

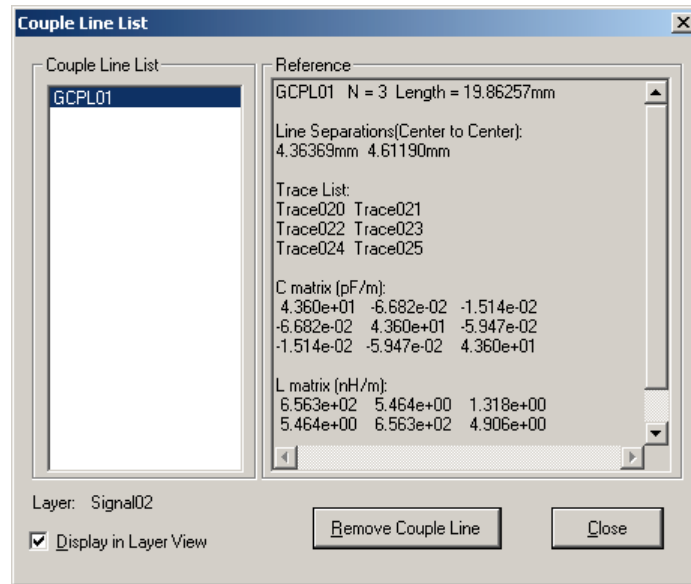
In the following cases, an error message box will appear

- If a trace crossed by line B is not a continuation of any of the coupled traces crossed by line A.
- If a trace has branches or stubs in the coupled section.
- If any adjacent traces are drastically not parallel to each other.
- If any member trace segments do not share the same “reference planes”.

**► To view and remove coupled lines:**

1. Click the Edit:Couple Line:List from the menu bar. A Couple Line List dialog box will appear. The number of coupled lines, parallel length, and average separations between lines will be shown in the Reference box. The name of the signal layer that contains the couple line is shown in the Layer area. The current couple

line will be highlighted in the Layer View window in yellow color if it is in the active layer.



#### NOTE:

Electrical properties of coupled lines listed in the Reference area of the Couple Line List dialog box are computed by a Method of Moment based electromagnetic field solver. In computation, metal skin effect loss, dielectric loss and dispersion, may also be taken into account.

2. Click the *Remove* button if you want the currently highlighted coupled line to become uncoupled. Otherwise, click *Close* to exit the Couple Line List dialog box.

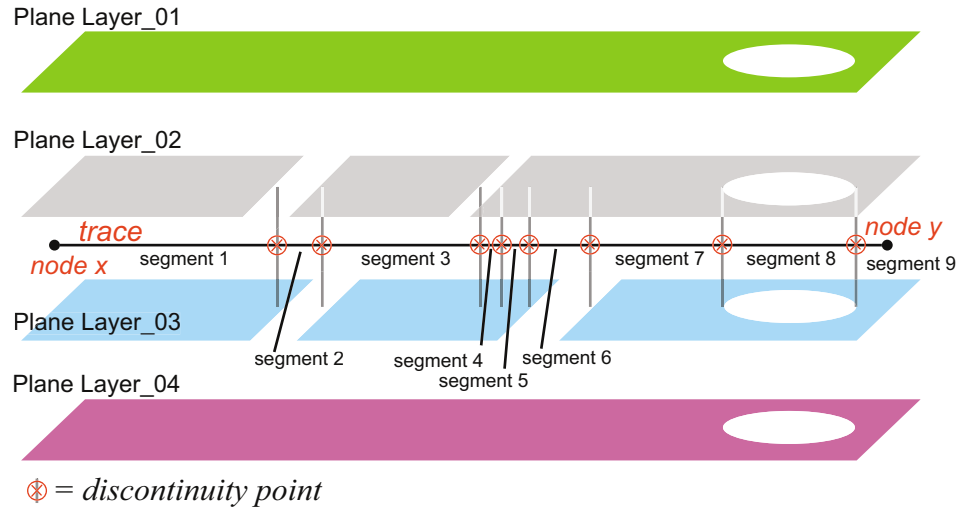
#### NOTE:

Whenever a trace becomes a part of a coupled line, it cannot be deleted or moved. Internal nodes of coupled lines cannot be connected to other components or to circuits.

### Traces Over “Split Planes”

The environment of a signal propagating along a trace is determined, in part, by the relationship of the trace to its reference plane(s). The horizontal separation (also called a gap) between planes on the same layer is typically referred to as a split. Planes may also contain voids in the form of holes. Traces which cross

these kinds of voids are subject to various electromagnetic phenomenon — which SPEED2000 models. Some illustrative details of how we model traces over split planes are provided in this section.



In SPEED2000, the computational environment along each trace is analyzed. If the upper reference plane and/or the lower reference plane are not identical at any point along a trace, the trace is deemed to “cross reference planes”. For example, this can happen when a trace crosses over a “split” or other void in an adjacent plane. Traces with such discontinuities are automatically divided into segments until each segment has a uniform computational environment. At the point where the discontinuity occurs, special algorithms are applied to take into account various effects. The figure above illustrates how a trace is segmented (in SPEED2000) due to different computational environments. The splits and voids in the planes adjacent to the trace cause 9 segments to be modeled. Note, the *Trace Over Split Plane* algorithms are applied only at the points of discontinuity.

#### NOTE:

A shape on a signal layer is referred to as a “patch”. Traces are “clipped” (sections deleted) when they overlap shapes on signal layers — if the trace clipping function is run. We recommend the .spd file is fully preconditioned, using the *shape process* button, on the Shape Toolbar, before each simulation or erroneous simulation results may occur. For more information on preconditioning the .spd file refer to [“Pre-Conditioning the .spd File For Simulation” on page 4-55](#).



When a layer stackup has several reference planes, the following rules determine which reference plane(s) is/are associated with each trace segment:

- For a trace segment above all planes in the package, its reference plane is the metal shape vertically and immediately under it;
- For a trace segment below all planes in the package, its reference plane is the metal shape vertically and immediately above it.
- For a trace segment between plane layers, it has two reference planes and they are the metal shapes vertically and immediately under and above the trace segment.

Two instances will cause SPDSIM to issue a warning message:

1. A trace segment without any reference plane. In the figure above, segment 8 has no metal shape vertically and immediately under and above the trace segment — thus, it has no reference plane. The user can either: ignore the warning — SPEED2000 will choose the closest adjacent metal layer as its reference (see figure); or, inspect the physical placement of the trace, and make minor modifications of either the trace or the reference plane shape so that the problematic trace segment has a reference plane vertically above or below the trace segment.
2. There are not sufficient mesh points near the point of trace discontinuity for the application of *Trace Over Split Plane* algorithms. The user can either: increase the number of mesh points; or, inspect the physical placement of the trace, and make minor modifications of either the trace locations or reference plane shape.

#### NOTE:

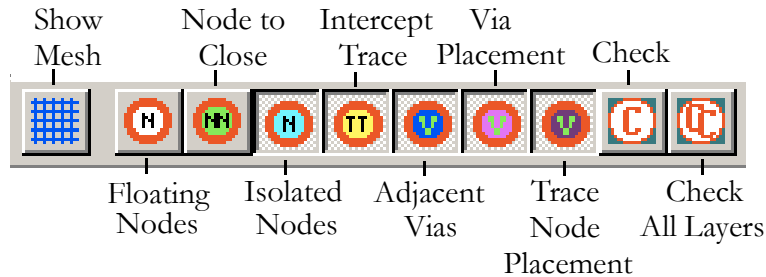
Increasing the number of mesh points significantly will result in larger computer resources necessary to do the simulation.

## Checking Drawing Errors

Some drawing errors can be hard to find. For example, you may think a trace is connected to a via since they appear to be overlapped with each other — but actually, the nodes of the trace and the via are not connected. By using the error checking tools of the software, you will easily spot those errors/warnings.

Before doing the error checking, you need to turn on the mesh first by clicking on the *Show Mesh* button. Whenever a drawing error is detected, an error/warning symbol will appear on the problematic area. These symbols will have an appearance as the buttons in the tool bar shown below.

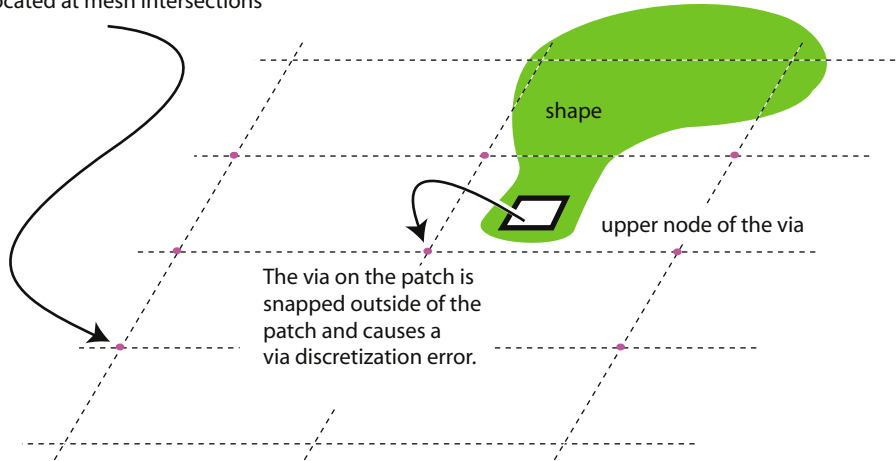
There are two types of warnings and five types of errors as shown below:



- Floating Nodes – Warning on nodes that have only one connection (with via, trace, patch) and they are not connected to a circuit.
- Node Too Close – Warning on two nodes that will be snapped to the same mesh point.
- Isolated Nodes – Error on nodes that do not connect to anything.
- Intercepted Traces – Error on two traces on the same signal layer that intercept with each other.
- Adjacent Vias – Error on two vias that are snapped to the same mesh point.
- Via Placement – Error on via placement. This error occurs when a via is located on a metal shape (in your model) but is virtually snapped to a mesh point (for numerical computations) outside the shape, or when a via is located outside a metal shape but virtually snapped to a mesh point inside the shape. (see figure) Our software will attempt to manage this type of discretization error internally — to complete an accurate computation; however, if it is unsuccessful an

error is reported. The solution is to increase the mesh or (if practical) move the via and shape components.

mesh points, for numerical discretization, are located at mesh intersections



- **Trace Node Placement** – Error on trace node placement. Occurs when a trace node is on a patch but snapped to a mesh point outside the patch, or when a trace node is outside a patch but snapped to a mesh point inside the patch. Also, our internal computation model requires that the node at the trace-patch junction is snapped to a mesh point located within the patch.

#### NOTE:

When dealing with Via Placement and Trace Node Placement problems, a finer mesh will often solve the problem.

#### ► To check the errors/warnings:

1. Depress the buttons for the kind of errors that you want to find: Floating Nodes, Node too Close, Isolated Nodes, Intercept Trace, Adjacent Vias, Via Placement and Trace Node Placement. Click the *Check* button on the tool bar. The checking process begins. The Check button (C) checks error on the active layer, while the CC button checks errors on all layers.
2. All errors found in the file will be marked with specific symbols — similar to the buttons in the tool bar shown above.



Check active layer button  
Check all layers button

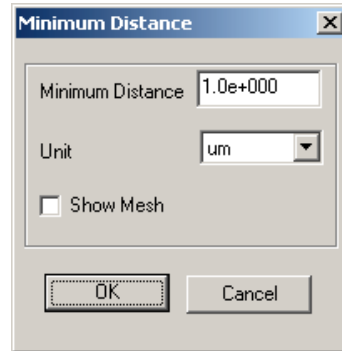
#### NOTE:

For a large file, a complete error checking process could take some time.

**► To establish error criteria/ minimum distance value:**

The minimum distance value is used by the adjacent node and via error checking routines.

1. Select Tools:Options>Error Criteria from the SPEED2000 menu bar. The Minimum Distance dialog box appears.



2. The default value is shown. Enter another value if desired and select the Units of interest.
3. Click on the Show Mesh option if you wish to view the numerical Mesh.
4. Click the OK button to exit the dialog box.

**Error Log File**

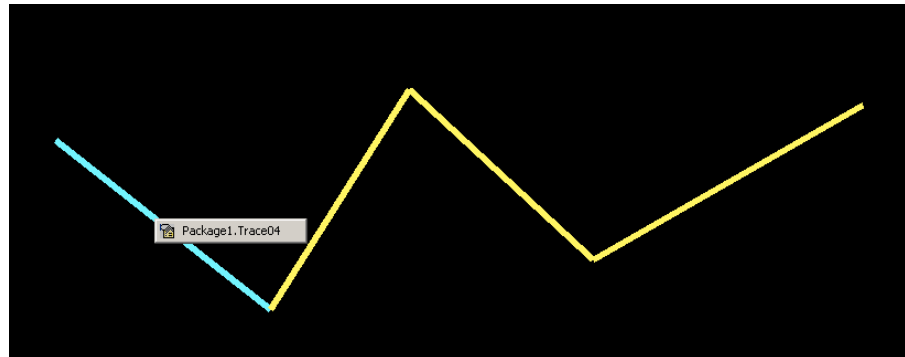
When you save a .spd file, all the error checking routines will be automatically activated to examine the file. If there are any errors present, an error log file will be generated with the name of xxx\_YYYY.err, where xxx is the name of the .spd file and YYYY is the name of the software module: GENERATOR, SIMULATOR or PowerSI. For more information on the error checking routines, refer to [“Checking Drawing Errors” on page 4-99](#).

## Modifying Object Properties

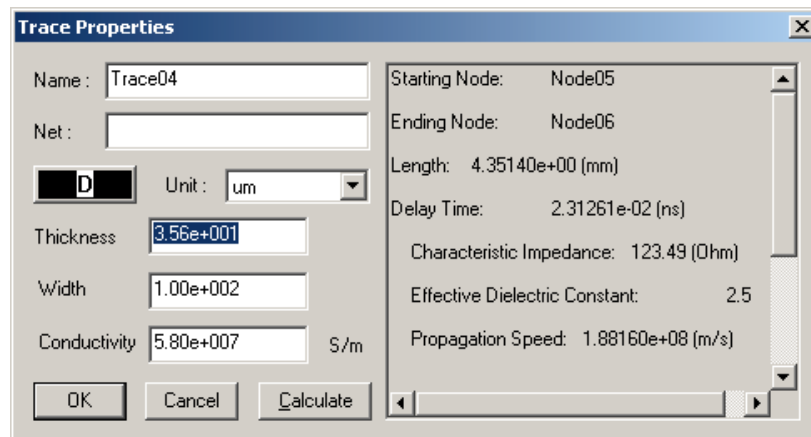
The properties of all package objects can be modified through their properties dialog box.

### ► To modify object properties:

1. Right-clicking on any selected object will cause a small pop-up dialog box, consisting of a short object ID report and the properties icon, to appear.



2. Click on the properties ICON and the properties dialog box for that object will appear. Editable attributes of objects include: Name, Net, Color, Unit, Thickness, Width, Connectivity, etc.



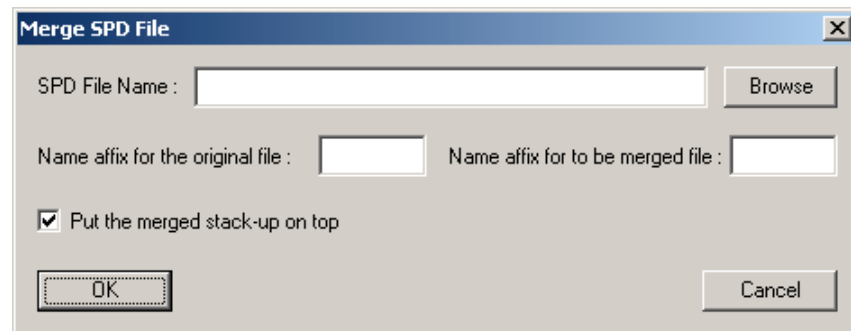
## Merging Two .spd Files

You can combine two .spd files together using the File:Merge SPD File function. This feature is useful when you want to simulate several structures together.

### ► To merge two .spd files:

1. Open the .spd file that you want to merge the other file into.

2. Select File:Merge SPD File from the menu bar. The Merge SPD File dialog box will appear.



3. Specify the name of the .spd file that you want to merge or you may use the Browse feature to find the file.

**NOTE:**

After the merge operation, the name of the resulting file will be the name of the original file. Select File:Save As if you want the resulting package to have another name.

4. If the merged structure is to be placed on top of the existing structure, check the box to **Put the merged stack-up on top**. Do not check the box if the merged structure is to be placed at the bottom of the existing structure.
5. To avoid duplication of object and net names, specify a name to affix to objects of the original file and another name to affix to objects of the merged file.
6. Click OK to merge the two files.

**NOTE:**

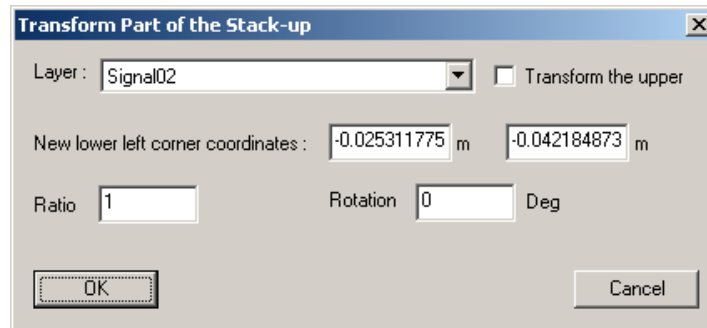
A dielectric layer (air) is automatically added between the two structures after merging two .spd files. You can change the properties of the layer if necessary. To specify the thickness of the air dielectric layer between the packages refer to [“To set properties of a dielectric \(medium\) layer:” on page 4-11](#).

### Moving and Rotating a Package

Use the **Transform Stack Up** and **Transform Stack Up By Nodes** features to move, scale and rotate layers of a package. These functions enable you to modify the position and size (if necessary) of a contiguous series of upper or lower layers. Use the [“To transform all \(or part\) of the stack-up:” on page 4-105](#) if you want to *move* by specifying a new lower left corner, *scale* based on a “ratio” and/or *rotate* based on degrees. Use the [“To transform the stack-up using nodes:” on page 4-106](#) if you want to automatically *move*, *scale* and/or *rotate* components based on nodes.

#### ► To transform all (or part) of the stack-up:

1. Click Edit:Transform Stackup from the menu bar, the Transform Part of the Stack-up dialog box appears.



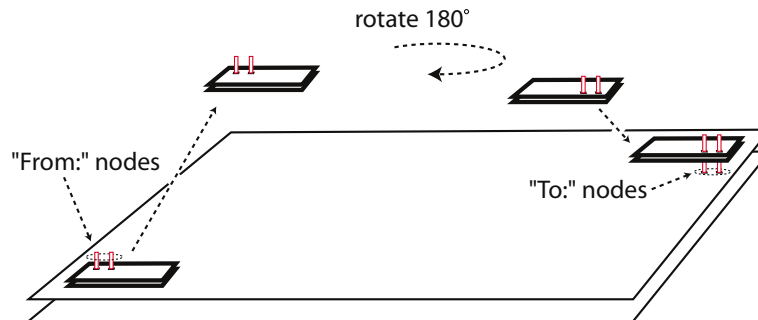
2. Choose a layer, from the Layer drop-down listing, and specify whether the upper layers starting from (including) that layer are to be transformed. For example, to transform an entire package (all layers) enter the bottom layer and place a check-mark in the **Transform the upper** option.
3. Enter the new x-and-y coordinates of the lower left corner of the transformed stackup. The default value is the current x-and-y coordinates of that point.
4. Enter any scaling factor needed to the **Ratio** field. For example, if the package needs to be scaled up 200 per cent, enter 2; and, if the package needs to be reduced 50 per cent, enter .5 for the **Ratio** value.
5. The stackup can also be rotated. The value of rotation is in degrees (with positive value for counter-clockwise). If the starting x-and-y coordinates values are not changed, the stack-up will be rotated, about its center, from the starting position.

#### NOTE:

A stack-up can be scaled, without being moved or rotated. A stack-up can be rotated, without being scaled or moved.

► **To transform the stack-up using nodes:**

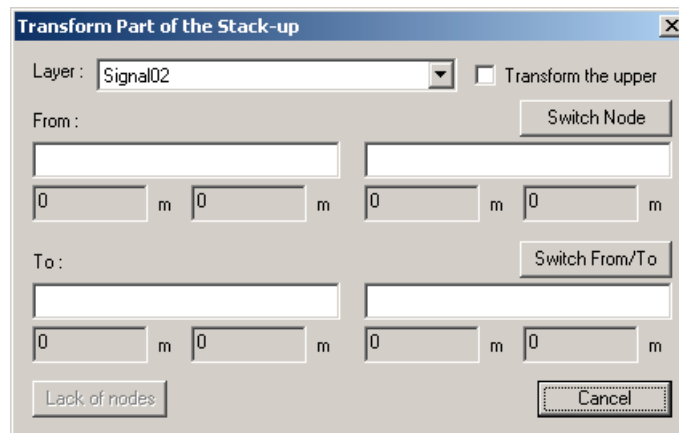
The use of this function requires that four related nodes in the stackup are “*known*”: two nodes must be specified for the “From:” fields — these nodes indicate the starting position; and two nodes must be specified for the “To:” fields — these nodes indicate the destination position. Any rotation or scaling that is required to place the transformed stackup into proper orientation will be handled by the function.



**NOTE:**

In order to use this feature you must either know the exact names of the nodes for entry to the 4 fields of the dialog box — or, consecutively select the nodes of interest within the package. Because this *apriori knowledge* is required to successfully use the feature, you may want to use the Find feature of SPDGEN first to familiarize yourself with the precise location and/or names of the nodes of interest.

1. Click Edit:Transform Stackup By Nodes from the menu bar, the Transform Part of the Stack-up dialog box appears.



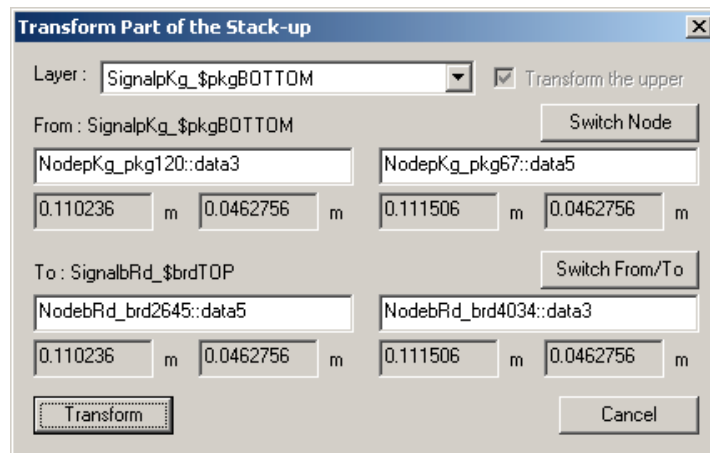


2. In the Layer Selection dialog box, close the view on all layers not involved in the transform procedure.
3. Uncheck the View Only Active Layer option.
4. Set the layer containing what will be the “From:” nodes to active.
5. Activate only the Select Node button of the Select Toolbar.
6. Turn on the Show Node and Auto Tip feature.
7. Zoom into the area that contains the “From:” nodes of interest.
8. Click the *Select* button of the Select Toolbar.
9. Click on the first “From:” node of interest, then hold down the shift key and click on the second “From:” node of interest.

**NOTE:**

The node selections you are making are going into a 4-member paste buffer that is utilized by the Transform by Nodes function.

10. Set the layer containing the “To:” nodes to active.
11. Zoom into the area that contains the “To:” nodes of interest.
12. Click the *select* button of the Select Toolbar.
13. Click on the first “To:” node of interest, then hold down the shift key and click on the second “To:” node of interest.
14. Select Edit>Transform Stackup By Nodes from the menubar. Observe the fields of the dialog box are populated with the four nodes you selected.



15. Click the Transform button. The components corresponding to the “From:” nodes will be transformed (moved, rotated, scaled) to align with the “To:” nodes.

**NOTE:**

To check the alignment, you can activate the Layer View window, select View>Net Manager. In the Net Manager dialog box deselect any nets not involved in the transformation. Select View 3D Window. Click the Hide unselected option to reveal the proximity of the components of the nets of interest.

16. To complete the electrical path that connects the now aligned nodes see [“To insert vias using existing vias:” on page 4-83](#) and [“To insert vias using existing package nodes and vias:” on page 4-85](#).

## Loading DXF Files as Background

If you have DXF files for your package structure, you can load these files into the software. The drawings on the screen from the DXF files are served as a background to facilitate your construction of the package structure with the software. If the scale and unit of the DXF file are known, you may also want to read the section [“Changing Scale and Unit before Loading a DXF File” on page 4-112](#).

### Loading a DXF File for the First Time

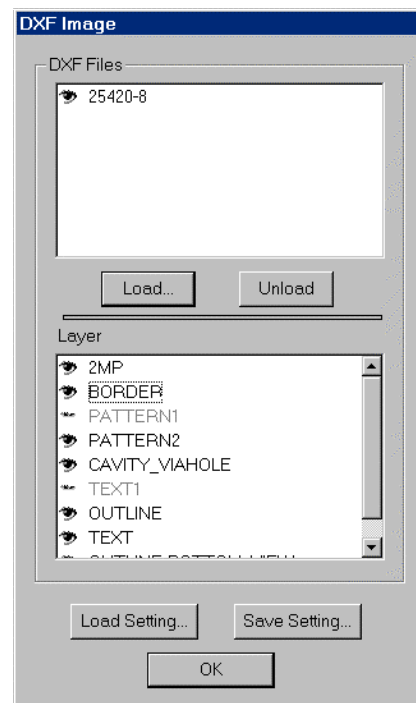
Assume you have a DXF file for one or all layers of a package. The first time you load a DXF file, two steps need to be followed:

1. Setting the dimension of the DXF image.
2. Setting the position of the DXF image.

Once the dimension and the position of the DXF image are set properly, you will not need to set the dimension and the position again next time you load this DXF file or load other DXF files for other layers of the package.

#### ► To read the DXF file:

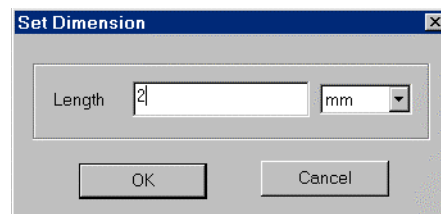
1. Click Background:DXF Image from the menu bar. The DXF Image dialog box appears.



2. Click the Load button in the DXF Image dialog box, an Open dialog box appears.
3. In the Open dialog box, browse the directory and select the file to be loaded. Click the Open button to exit the Open dialog box.
4. After the DXF file is read into the software, the name of the DXF file appears in the upper box in the DXF Image dialog box, while the names of graphics layers of the DXF file are listed in the lower box. The Image of the DXF file is drawn in the Layer View window. You can click on the DXF file name in the upper box to switch the view of the file between ON and OFF. For the DXF file highlighted in the upper box, the lower box lists graphics layers of the corresponding DXF file. You can click on the names of the DXF graphics layers to turn the display of these layers to ON or OFF.
5. You can load multiple DXF files and view them all at the same time. If you want to remove one DXF file, simply select the name of the file in the upper box of the DXF Image dialog box, and click the Unload button.
6. Click the OK button to exit the DXF Image dialog box.

► **To set the dimension:**

1. Click Background: Set Dimension from the menu bar.
2. Move the cursor to the Layer View window. Click the left mouse button once, then hold and drag the mouse. A square outline is drawn in the Layer View window. After releasing the left mouse button, a Set Dimension dialog box appears.
3. Enter a physical length in the Set Dimension dialog box. **The physical length that you entered corresponds to the length of the side of the square outline drawn in the Layer View window.**
4. Click the OK button to exit the Set Dimension dialog box.



**NOTE:**

If you want to re-set the dimension of the DXF image, simply follow the previous 4 steps again.

► **To set the position:**

1. Click Background: Set Position from the menu bar.
2. Select a location in the Layer View window by clicking the left mouse button. Then hold and drag the mouse cursor to another location, while a line connecting the two locations is drawn, and release the left mouse button.
3. The DXF image will then be displaced according to the displacements of the starting and the ending locations in the previous step.

**NOTE:**

If you want to re-position the DXF image, follow the previous 3 steps again.

If you load more DXF files now, images of those files will be drawn according to the settings of the dimension and the position for the first DXF file unless the dimension and the position are re-set.

**If different DXF files need different settings, or before you exit the software, we suggest to save the settings of dimensions and positions to avoid performing those settings each time you load a DXF file.**

► **To save the settings for dimension and position:**

1. Click the Save Setting button in the DXF Image dialog box. The Save As dialog box appears.
2. In the Save As dialog box, go to the proper directory you want the settings to be saved. Type the name of the file for the settings in the box to the right of File Name. Click the Save button to exit the Save As dialog box.

### ***Loading DXF Files Using Saved Settings of Dimension and Position***

If you want to load a DXF file with settings saved in a DXF setting file, load the setting file first.

► **To load the DXF setting file:**

1. Click Background:DXF Image from the menu bar. The DXF Image dialog box appears.
2. Click the Load Setting button in the DXF Image dialog box. The Open dialog box appears.
3. In the Open dialog box, browse the directory and select the DXF setting file (with the suffix of .cfg). Click the Open button to exit the Open dialog box.

Now if you load a DXF file by clicking the Load button in the DXF Image dialog box, the image of this newly loaded DXF file will be drawn using the dimension and the position specified in the DXF setting file. Even if you

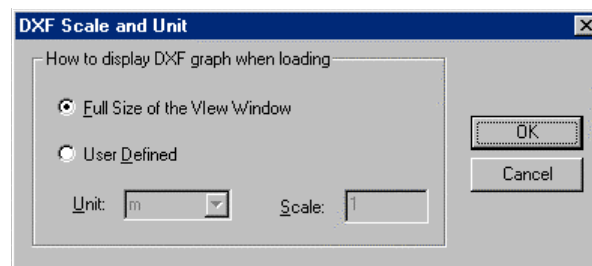
unload some DXF files now, the dimension and the position settings will be left unchanged so you can load other DXF files without worrying about the alignment of their DXF images.

### ***Changing Scale and Unit before Loading a DXF File***

If the scale and unit of a DXF file are known, these two factors can be preset so that the dimension does not need to be adjusted after loading the file.

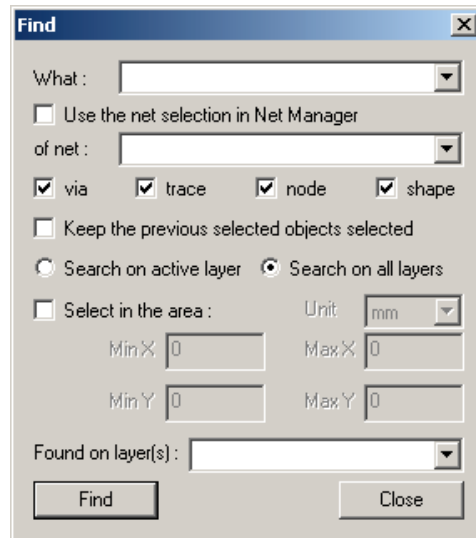
► **To set the scale and unit of DXF files for loading:**

1. Click Tools: Options: DXF Scale and Unit from the menu bar. The DXF Scale and Unit dialog box appears.
2. The default setting is to fit the DXF image in the Layer View window.
3. Select the User Defined option and enter the Unit and Scale parameter. If the Scale is set to 2, all data in the DXF file will be multiplied by 2 and then drawn on the screen.



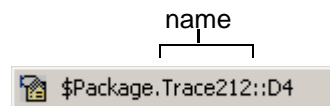
## Finding Objects

The Find dialog box enables you to find objects in the .spd file on the basis of explicit object [names](#) and/or “[wildcards](#)”. It functions with the Net Manager and Layer Selection dialog boxes to constrain searches and make them efficient.



### Object Names and Wildcards

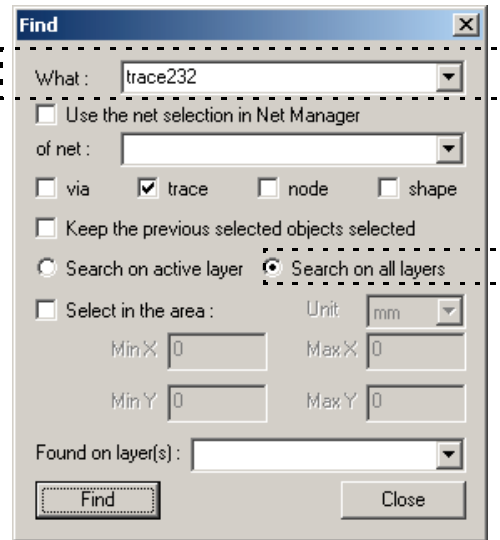
A “wildcard” is an asterisk [\*] or question mark [?] character inserted to the search criteria. The \* character inserted to the search criteria indicates: find all objects with the specified characters and any other characters — at the position indicated — as a part of their name. The ? character inserted to the search criteria indicates: find all objects with the specified characters and any other one character at the location indicated by the ? character. Some examples: The entry *Via\** will retrieve all objects with the letters v-i-a (in sequence) in their name. The entry *Via\*2* will retrieve all objects with the letters v-i-a (in sequence) and ‘2’ in their name. The entry *Via\*1?2* will retrieve all objects with the letters v-i-a (in sequence), and a ‘1’, and a ‘2’, separated by any one other character in their name at that position specified — that is: Via152, Via112, Via142.



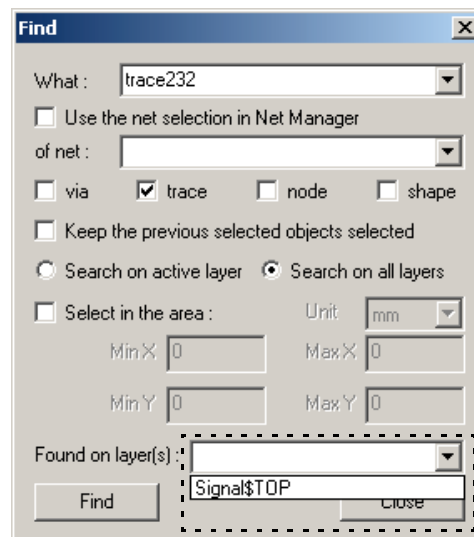
The “names” of objects are confined to the characters present in the .spd file syntax. Thus, entering *\*age\** with an assumption this will retrieve objects such as Package1.trace212, Package1.via04, etc. ... will not work. For, in fact, this information, which is displayed on the SPEED2000 GUI, is a construct and not the actual name of the object *within the .spd file*. For detailed information on object naming conventions, refer to the [.spd File Format Reference](#) document.

► **To find objects on the basis of their object names:**

1. To open the Find dialog box, click on Edit:Find in the menu bar. The Find dialog box appears.
2. Enter the object's name to the **What:** field of the Find dialog box.

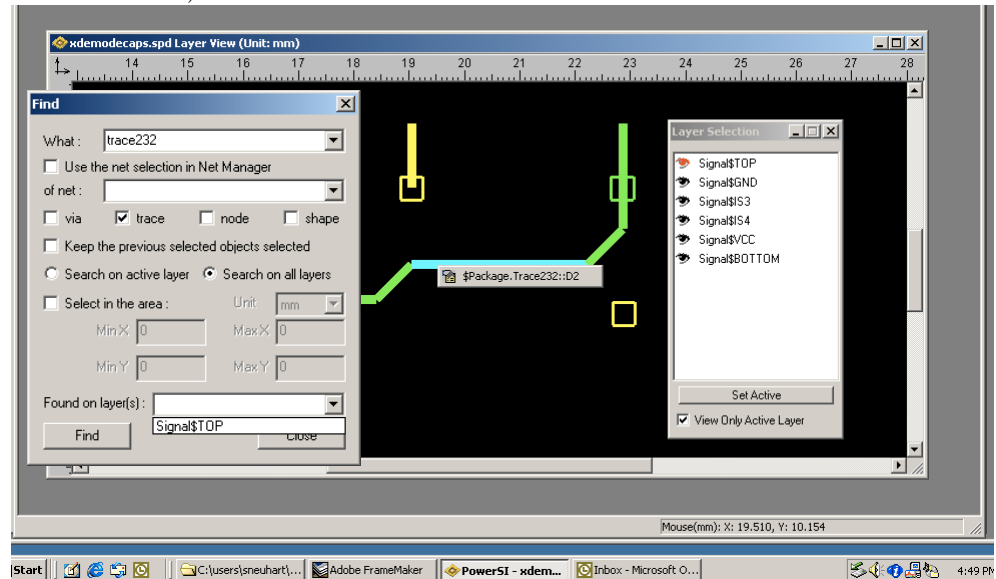


3. Click the button next to **Search on all layers** to search all layers of the package for the object that meets the search criteria.
4. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
5. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.





6. Select any layer shown in the drop-down listing to immediately make it active and view the object.



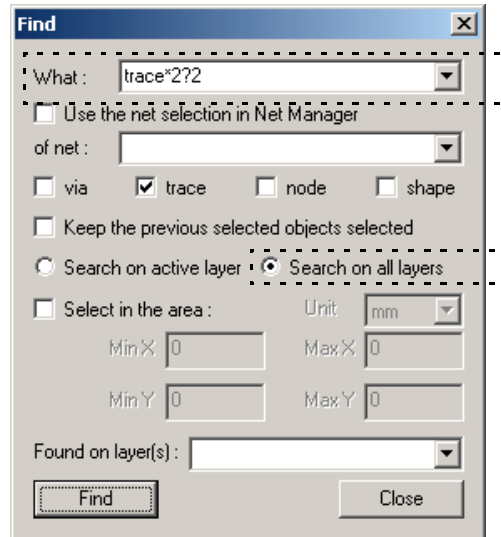
7. Select the **View Only Active Layer** option of the Layer Selection dialog box to view only the “active layer”.



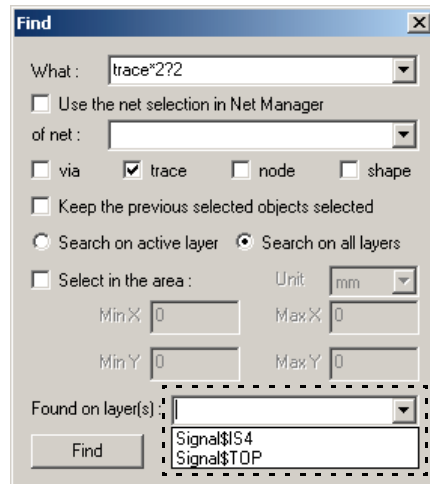
8. Click the *Close* button to exit the Find dialog box.

► **To find objects on the basis of patterns:**

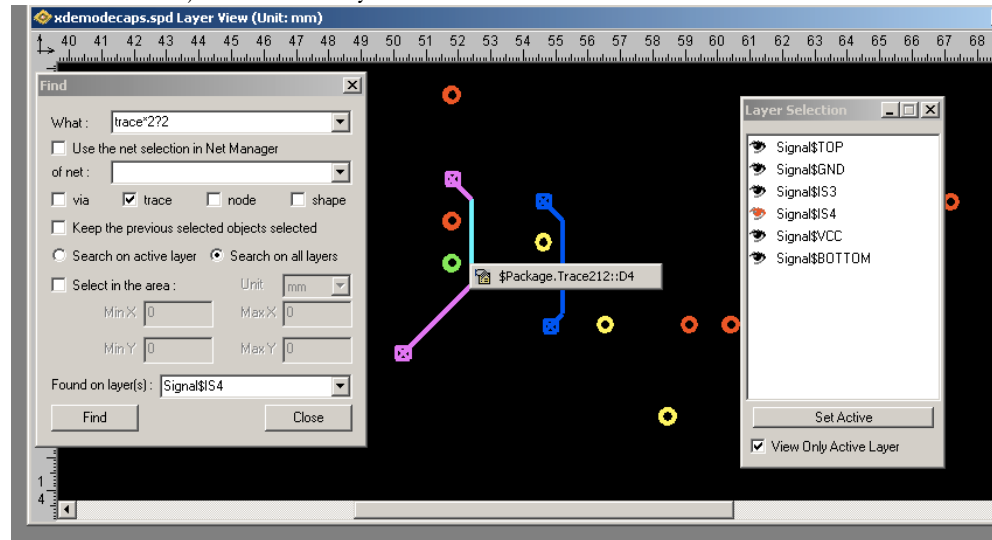
1. To open the Find dialog box, click on Edit:Find in the menu bar. The Find dialog box appears.
2. Enter as much of the object's name that you may know for certain to the **What:** field of the Find dialog box. Enter “wildcards” for characters that you are uncertain of. For example, we entered *trace\*2??* below. Note, that \* and ? are both wildcards. Refer to [“Object Names and Wildcards” on page 4-113](#) for more information on wildcard characters and how they work.



3. Click the button next to **Search on all layers** to search all layers of the package for any objects that meet the search criteria.
4. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
5. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.



6. Select any layer shown in the drop-down listing to immediately make it active and view the objects on that layer.



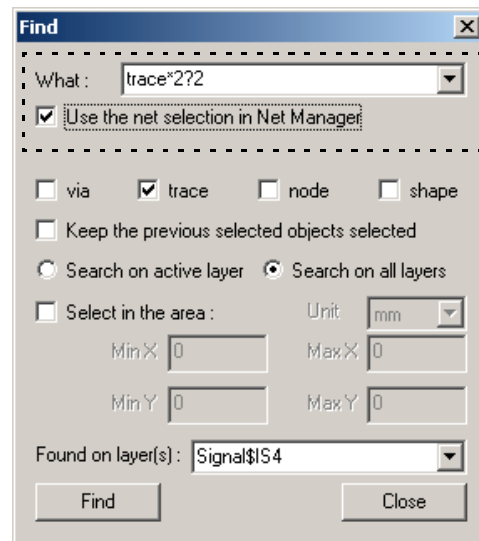
7. Select the **View Only Active Layer** option of the Layer Selection dialog box to view only the “active layer”.



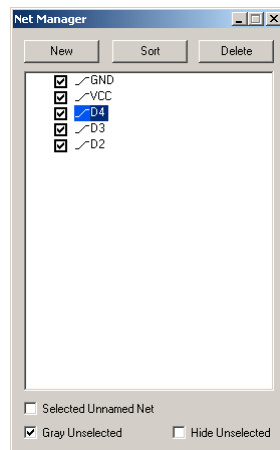
8. Click the *Close* button to exit the Find dialog box.

► **To find objects using the Net Manager:**

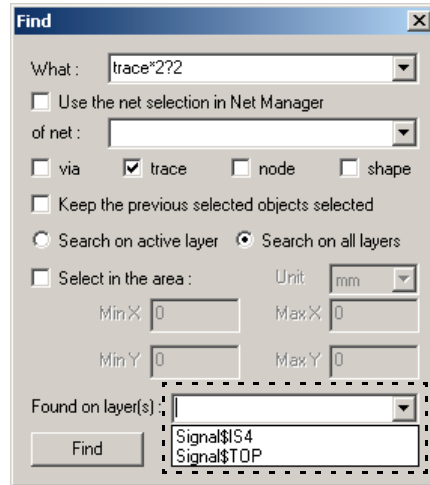
1. To open the Find dialog box, click on Edit:Find in the menu bar. The Find dialog box appears.
2. Open the Net Manager dialog box. Click on View:Net Manager, the Net Manager dialog box appears. For details of how to use the Net Manager, refer to [“Net Management” on page 4-15](#).
3. Enter as much of the object's name that you may know for certain to the **What:** field of the Find dialog box. Enter “wildcards” for characters that you are uncertain of. For example, we entered *trace\*2?2* below. Note, that \* and ? are both wildcards. Refer to [“Object Names and Wildcards” on page 4-113](#) for more information on wildcard characters and how they work.



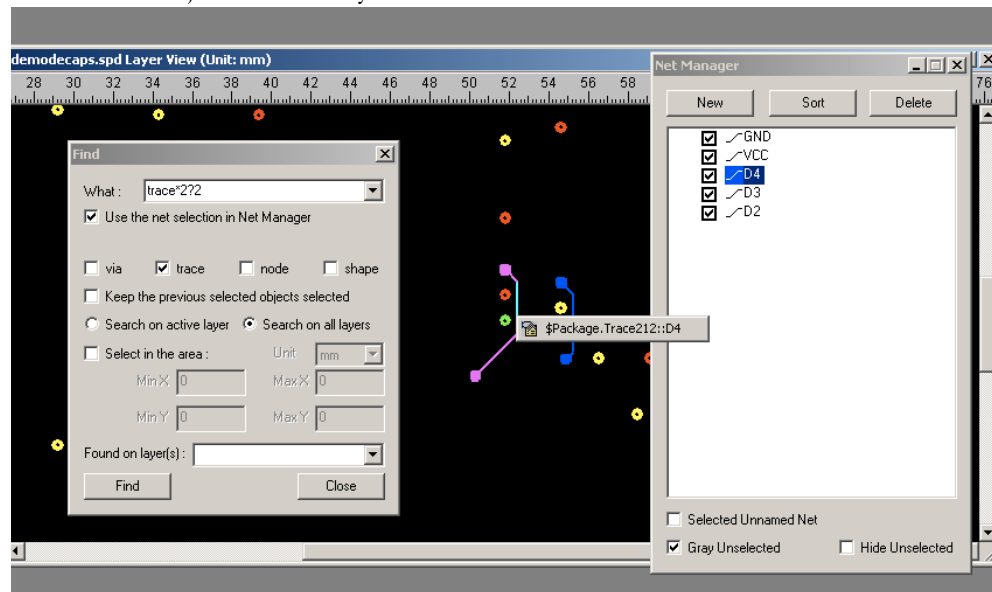
4. Click the box next to **Use the net selection in Net Manager**, a checkmark appears and the **of net:** option disappears.
5. Click on the net of interest, in the Net Manager dialog box, to highlight it as shown.



6. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
7. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.



8. Select any layer shown in the drop-down listing to immediately make it active and view the objects on that layer.



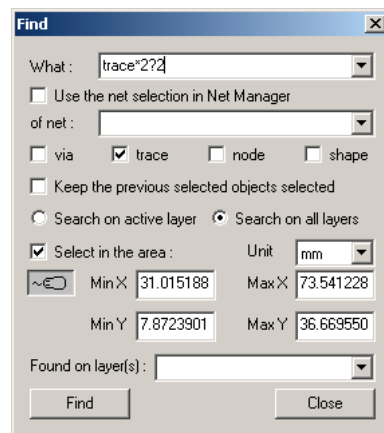
9. If necessary, select the **View Only Active Layer** option of the Layer Selection dialog box to view only the “active layer”.



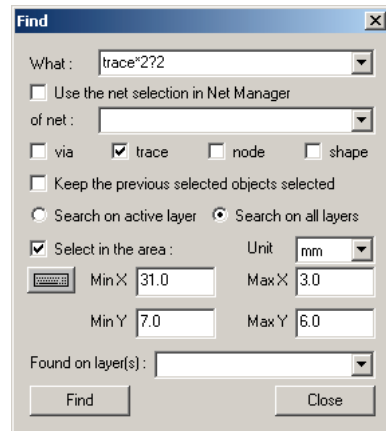
10. Click the *Close* button to exit the Find dialog box.

► **To search for objects in a selected area:**

1. To open the Find dialog box, click on Edit:Find in the menu bar. The Find dialog box appears.
2. Enter the object's name, if you know it for certain, to the **What:** field of the Find dialog box. If you are uncertain of the objects name, refer to [“To find objects on the basis of patterns:” on page 4-116](#).
3. Click the button next to **Search on all layers** to search all layers of the package for any objects that meet the search criteria.
4. Make certain the area of interest is viewable in the Layer View window.
5. Click the box next to **Select in the area:** option of the Find dialog box, the fields associated with the select function are highlighted and a mouse icon is viewable. Drag the mouse, while holding down the left mouse button, to select the area of interest.. The fields of the X- and Y- coordinates are filled with values to indicate the area selected.



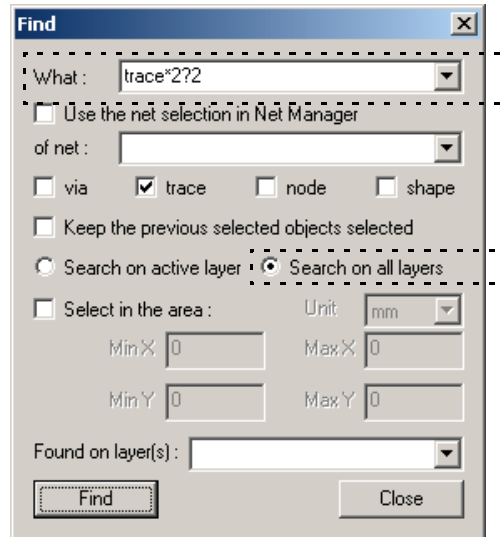
6. If necessary, you may adjust the values manually. Click the box with the mouse icon in it, a keyboard icon appears. Edit the values as necessary.



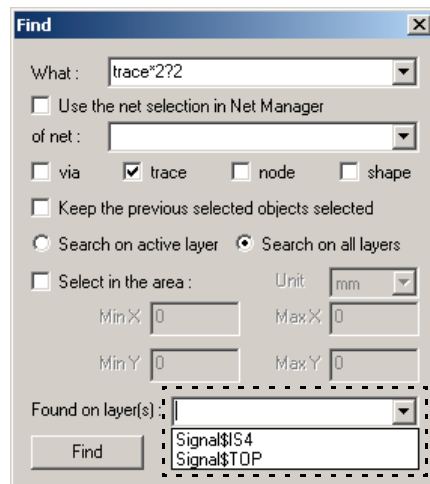
7. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
8. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.
9. Select any layer shown in the drop-down listing to immediately make it active and view the objects on that layer.
10. Select the **View Only Active Layer** option of the Layer Selection dialog box to view only the “active layer”.
11. Click the *Close* button to exit the Find dialog box.

► **To combine search results:**

1. To open the Find dialog box, click on Edit:Find in the menu bar. The Find dialog box appears.
2. For the first search, enter as much of the object's name that you may know for certain to the **What:** field of the Find dialog box. Enter "wildcards" for characters that you are uncertain of. For example, we entered *trace\*2?2* below. Note, that \* and ? are both wildcards. Refer to ["Object Names and Wildcards" on page 4-113](#) for more information on wildcard characters and how they work.

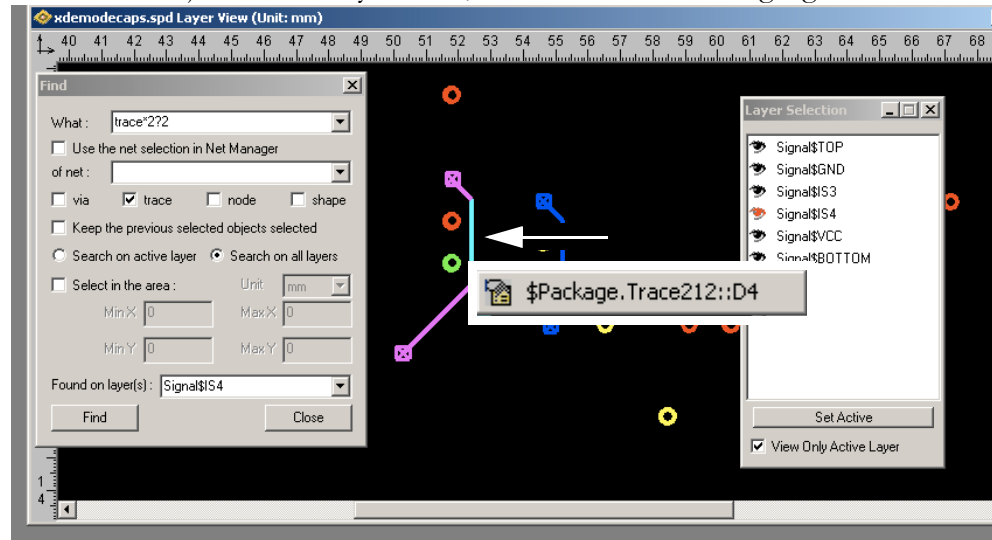


3. Click the button next to **Search on all layers** to search all layers of the package for any objects that meet the search criteria.
4. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
5. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.





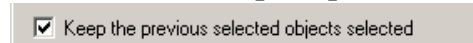
6. Select any layer shown in the drop-down listing to immediately make it active and view the objects on that layer. Note, that Trace212 trace is highlighted.



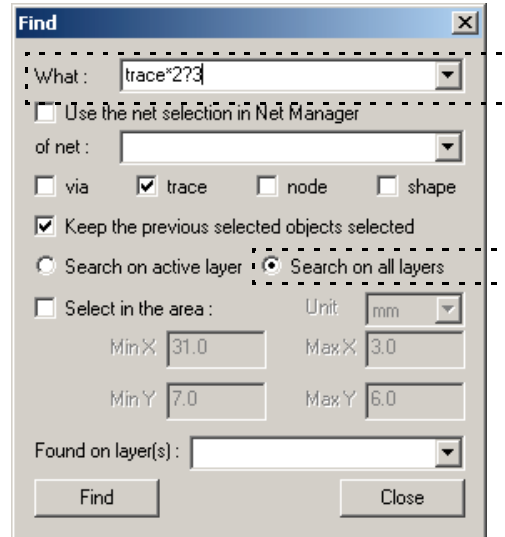
7. Select the **View Only Active Layer** option of the Layer Selection dialog box to view only the “active layer”.



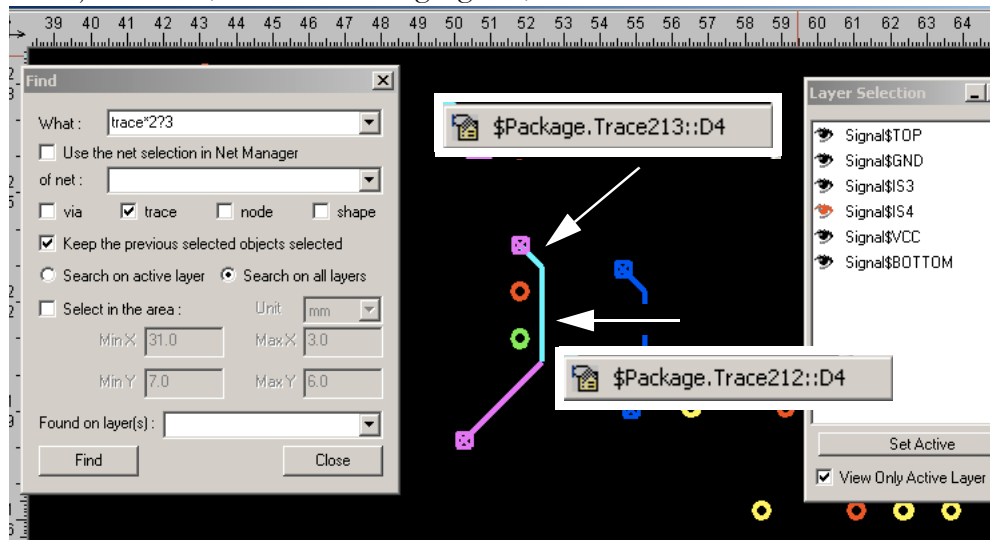
8. Click the box to **Keep the previous selected objects selected**.



9. For the second search, again enter as much of the object's name that you may know for certain to the **What:** field of the Find dialog box. For example, we entered *trace\*2??* below.



10. Click the button next to **Search on all layers** to search all layers of the package for any objects that meet the search criteria.
11. Click the **Find** button — the objects found (if any) will be highlighted in the Layer View window.
12. Click the **Found On Layer(s):** drop-down listing to view a list of the layers that contain the object.
13. Select any layer shown in the drop-down listing to immediately make it active and view the objects on that layer. Note, that because we “kept” the previous objects found, two traces are highlighted, Trace213 and Trace212.



14. Click the *Close* button to exit the Find dialog box.

## Working with Circuits

A collection of electronic components that have been assembled and interconnected to perform a given function is commonly referred to as a circuit. In Sigrity software, a new term, **partial circuit**, is introduced. A “partial circuit” has the following characteristics:

- A partial circuit is a circuit which does not directly connect to any other circuits (or partial circuits).
- A partial circuit must be directly connected to a package.

A partial circuit is described by a **partial circuit definition**. Distinct partial circuits can share a common partial circuit definition. For example, partial circuit A and partial circuit B are two unique partial circuits, but the topology and components of both partial circuits are identical. Thus, partial circuit A and partial circuit B can share a partial circuit *definition* X. But, even if they share a logical definition, partial circuit A and partial circuit B may connect to a package at different physical locations. For instance, partial circuit A may attach nodes x1 and x2 to certain physical nodes of the package, whereas partial circuit B may attach nodes x1, x2, x3 and x4 to other physical nodes of the package.

### NOTE:

You can add partial circuits using a common partial circuit definition — but, if you later modify the common partial circuit definition, all circuits using that definition will be affected.

The syntax of a partial circuit definition is similar to that of SPICE. Detailed descriptions of the syntax for partial circuit definitions within .spd files can be found in the Sigrity [.spd File Format Reference](#) document. Partial circuit components currently modeled for SPEED2000 include:

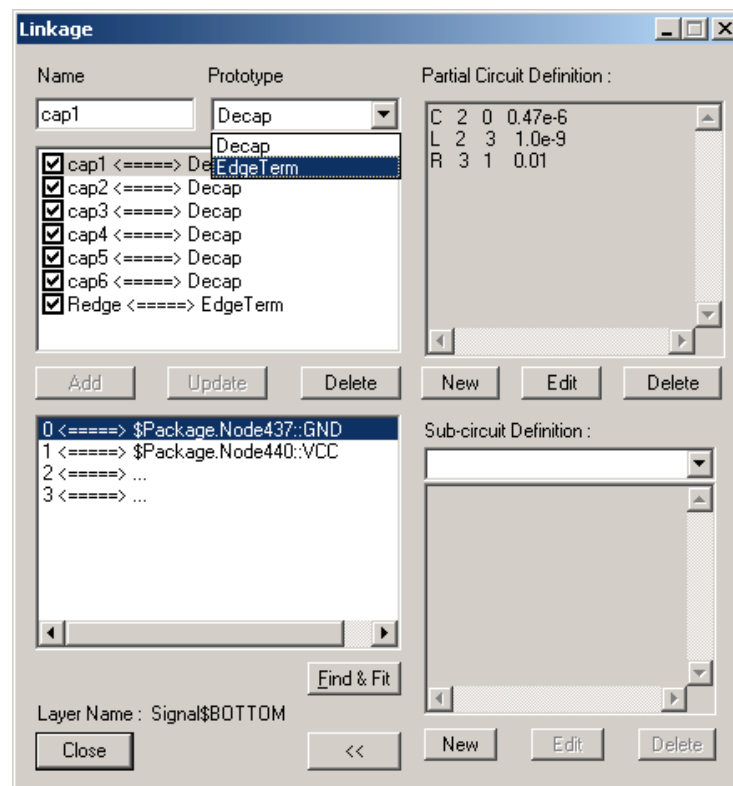
- resistors
- inductors
- capacitors
- current source
- voltage source
- transmission line
- voltage controlled voltage source

- current controlled current source
- voltage controlled resistor
- voltage controlled current source
- current controlled voltage source
- mutual capacitor matrix
- mutual inductor
- mutual inductor matrix
- and buffers defined by I/O Buffer Information Specification (IBIS)

You may also use “subcircuit” definitions. Again, each instance of a subcircuit description must be preceded by a **sub-circuit definition**. For detailed information on the .spd file syntax and how subcircuits are defined and used refer to the [.spd File Format Reference](#) document (available on our website).

### Overview Of The Linkage Dialog Box

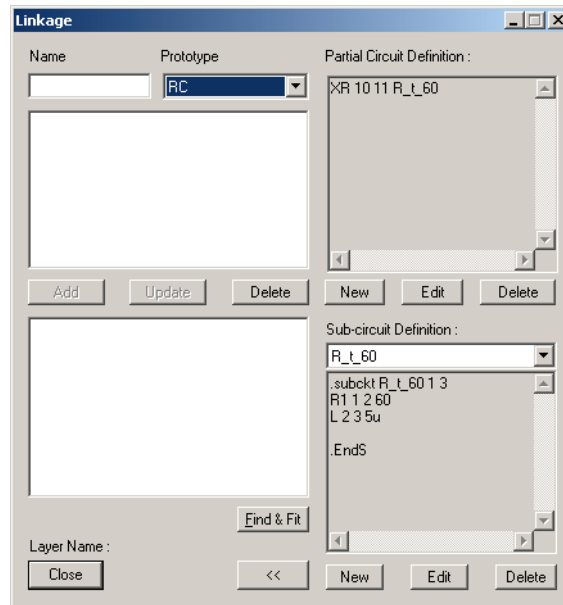
The Linkage dialog box contains several important areas to accomplish the tasks associated with managing circuits. Select Setup: Linkage from the menu bar to display the Linkage dialog box. The Name field is used to name the unique



partial circuit instance. The Prototype field lists the partial circuit prototype definitions that are available — as shown in the figure they are “Decap” and “EdgeTerm”. The Partial Circuit Definition area displays the component descriptions (in .spd syntax) for a defined circuit prototype. As shown in the figure, Partial Circuit Definition “Decap” consists of a capacitor ‘C’, an inductor ‘L’, and a resistor ‘R’. The circuit nodes of C are 2 and 0. The value of capacitor C is 0.47e-6 uF. The circuit nodes of inductor, L are 2 and 3. The value of inductor L is 1.0e-9 nH. The circuit nodes of R are 3 and 1. The value of resistor R is 0.01  $\Omega$ . The upper left portion of the dialog box displays a listing of all of the partial circuits defined for the package. Each line reports the partial circuit name and the Prototype (Partial Circuit Definition) used, in the form: circuit-Name <=====> Prototype. For example in the figure shown, cap4 <=====> Decap, indicates that the partial circuit “cap4” uses the Decap prototype definition. Note that cap1, cap2, cap3, cap5 and cap6 also use this prototype — whereas, the circuit Redge uses the EdgeTerm prototype.

The lower left portion of the dialog box reports all of the package linkages for the partial circuit highlighted in the upper window. As shown in the previous figure, the partial circuit “cap1” contains 4 circuit nodes (0, 1, 2, 3) in its topology. Some of these nodes are connected to package nodes as reported in the lower Linkage dialog box window. As shown in the figure, each line reports the circuit node name and the physical package node name, in the form xxx<=====> yyy, where xxx is the circuit node name and yyy is the package node name. Specifically, in .spd syntax, \$Package.Node437::GND, provides the x- and y- coordinates of placement and the Layer Name field (highlighted below as Signal\$BOTTOM) indicates the vertical (z-coordinate) placement in the stackup. If a circuit node is shown connected to “...”, instead of a specific pack-

age node, this indicates that no package node has been linked to this circuit node.



If a subcircuit component is contained in the .spd file, it will be displayed in the Subcircuit prototype definition area (lower right) of the Linkage dialog box. To link subcircuit nodes to the package, you must identify the partial circuit definition which uses that subcircuit definition by selecting it in the Prototype field. Then, the procedure is the same as for Partial Circuit nodes (refer to [“To make a linkage:” on page 5-9](#)).

The Linkage dialog box has several buttons. The << or >> button toggles the Linkage dialog box to expand to display additional information and features or contract to conserve space. The Add button highlights when a new circuit Name is entered. It refreshes the circuit listing window with the name of the new circuit. The Update button highlights when changes affecting the circuit are entered. It refreshes the circuit listing window with the most current information. There are two delete buttons on the Linkage dialog box. The Delete button (under the circuit listing window) is used to delete circuits. The Delete button (under the Partial Circuit Definition window) is used to delete circuit definitions. The New button is used to add new partial circuit definitions. It causes the New Partial Circuit Definition dialog box to appear. The Edit button is used to edit partial circuit definitions. It causes the New Partial Circuit Definition dialog box to appear and it is populated with the circuit definition specified in the Prototype field. And finally, selecting a circuit then clicking on the Find & Fit button finds and displays the circuit in the Layer View window. Use

the check box, ☒ , that precedes each circuit in the Linkage dialog box to conveniently select and de-select circuits for simulation.

## Adding New Partial Circuits

### ► To add new partial circuits:

1. Click Setup: Linkage from the menu bar. The Linkage dialog box appears. If the dialog box is contracted, click the Expand button (>>) to view the Partial Circuit Definition area. If you loaded an existing .ckt (or .spd) file, you may have several partial circuit definitions in the Prototype list box of the Linkage dialog box to select from — otherwise, you will need to define the circuit components, using the New button of the Linkage dialog box. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) for a detailed overall review of Linkage dialog box.
2. All partial circuits are based on partial circuit definitions (prototypes). To add a new partial circuit — and an appropriate prototype does not exist — first, refer to [“Creating New Partial Circuit Definitions” on page 5-5](#); then, return to these instructions.
3. To add a new circuit — based on an existing Partial Circuit Definition — type in the name of the new partial circuit in the Name field (of the Linkage dialog box), select a partial circuit definition in the Prototype list box, then click the Add button of the Linkage dialog box. You will then have a new partial circuit specified by that particular partial circuit definition. Observe your new circuit displayed in the upper window’s circuit listing. Each line reports the partial circuit name and the Prototype (Partial Circuit Definition) used, in the form:

circuitName <=====> Prototype

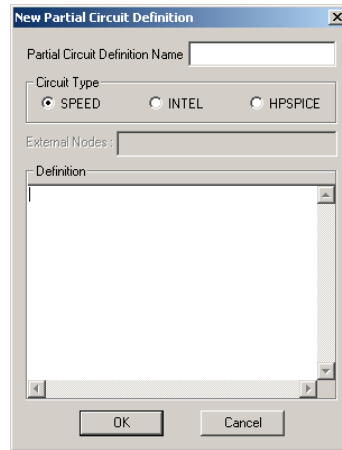
## Creating New Partial Circuit Definitions

Sometimes you need to add a partial circuit and an appropriate existing Partial Circuit Definition does not exist. In this case you will want to add a new partial circuit definition.

### ► To add a new partial circuit definition:

1. Click Setup: Linkage from the menu bar. The Linkage dialog box appears. If the dialog box is contracted, click the Expand button (>>) to view the Partial Circuit Definition area.

2. Click the New button of the Linkage dialog box. The New Partial Circuit Definition dialog box appears. You may enter the components and topology of the



new partial circuit in the Definition window. Use the syntax and statement definitions provided in the Sigrity [.spd File Format Reference](#) document. Or, if you already have a SPICE file, you can open it using Notepad or any window based text editor. Highlight the portion of your SPICE file you want to import. Press Ctrl-C, or click Edit: Copy. Go back to the New Partial Circuit Definition dialog box, click the place where you want your text to be placed in the Definition field, and press Ctrl-V.

3. For Circuit Type click the button next to SPEED. The Circuit Type, in the general case, should always be SPEED. The other options are for future enhancements.
4. Assign a name to the new partial circuit definition using the Partial Circuit Definition Name field, then click OK. This definition will then be available in the Prototype name list of the Linkage dialog box.

## Loading an External Partial Circuit Definition File

The .spd partial circuit library contains our partial circuit definitions. The file name of the partial circuit library ends with .ckt extension; or, you can use the partial circuit definitions defined in another .spd file.

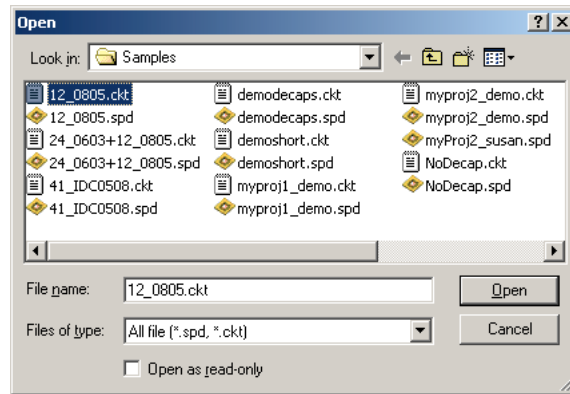
### NOTE:

When you load external circuit file information, any circuit definitions that already exist (same name and same contents) will not be imported again. If the new circuit definition has the same name as an existing one, but different contents, the new circuit definition will be renamed with the addition of an extension.



► **To load external partial circuit definitions from a .spd or .cur file:**

1. Click File:Merge Circuit Definitions. The Open dialog box appears.



2. Enter the circuit file name. You can select a .ckt file or an .spd file. If a .spd file is selected, only its circuit definition portion will be extracted to the target .spd file.
3. Click Open to load the circuit definitions.

## Deleting a Partial Circuit

► **To delete a partial circuit:**

1. Click Setup: Linkage from the menu bar. The Linkage dialog box appears. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. In the upper window of the Linkage dialog box, select the partial circuit of interest by clicking the left mouse button.
3. Click the Delete button under the partial circuit listing.

**NOTE:**

When a partial circuit is deleted, all of the connections between the circuit nodes and the package nodes are deleted.

## Deleting a Partial Circuit Definition

► **To delete a partial circuit definition:**

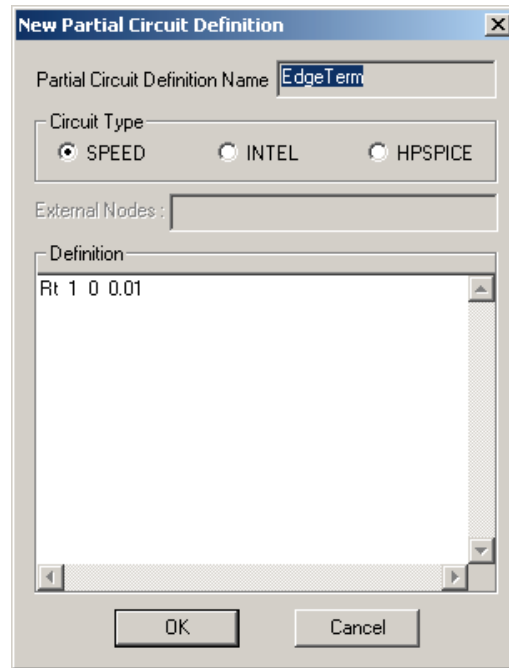
1. Click Setup: Linkage from the menu bar. The Linkage dialog box appears. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. Click the Expand (>>) button in the Linkage dialog box, the components of the partial circuit definition which is highlighted in the Prototype list will appear in the window of the extended portion of the Linkage dialog.
3. Click to highlight the partial circuit definition of interest from the Prototype listing.

4. Click the *Delete* button under the Partial Circuit Definition field, a confirmation notice will appear. This notice will remind you about the circuits that will be deleted along with the definition.
5. Click Yes if you want to proceed.

## Editing a Partial Circuit Definition

### ► To modify a partial circuit definition:

1. Click Setup: Linkage from the menu bar. The Linkage dialog box appears. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. Click the Expand (>>) button in the Linkage dialog box, the components of the partial circuit definition which is highlighted in the Prototype list will appear in the window of the extended portion of the Linkage dialog.
3. Click to highlight the partial circuit definition of interest from the Prototype listing.
4. Click the Edit button under the Partial Circuit Definition field. The New Partial Circuit Definition dialog box will appear. This dialog box will enable you to do text editing on the selected partial circuit definition. You can either type in the modifications or use Ctrl-C, Ctrl-V to copy and paste texts from other existing text files. After the modification, click the OK button to exit the Partial Circuit Definition dialog box.
5. If the partial circuit definition is in use, a confirmation notice will appear. This notice will remind you about the circuits that will be modified along with the definition.
6. Click Yes if you wish to proceed.

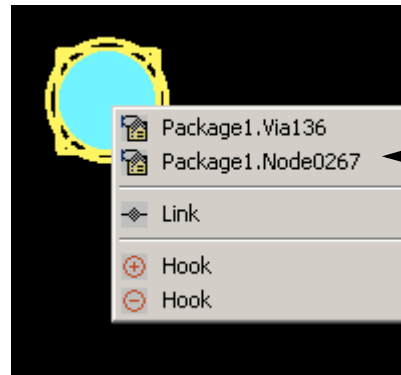


## Linking Partial Circuits to the Package

Once you have built package and defined circuits, the next step is to specify connections between circuits and the package.

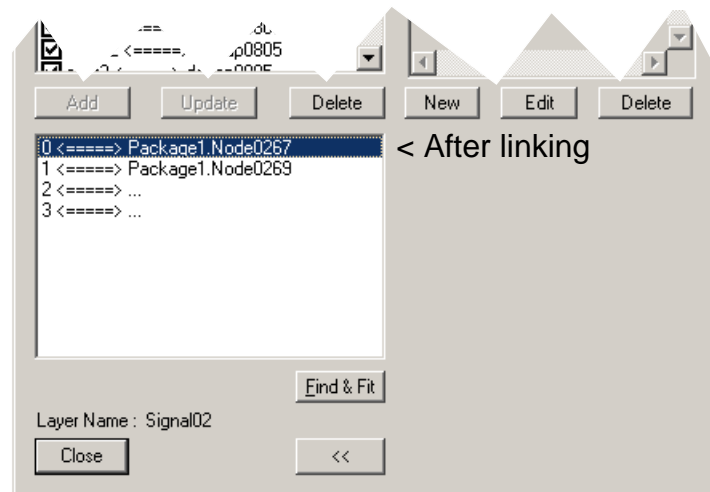
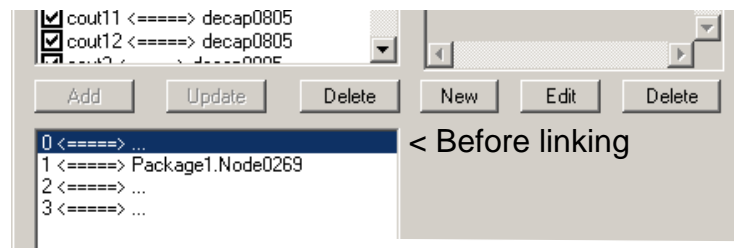
### ► To make a linkage:

1. Open the Linkage dialog box by clicking Setup:Linkage from the menu bar. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. In the Linkage dialog box, highlight the line (in the upper circuit listing) that contains the circuit you want to select by clicking the left mouse button at the line.
3. In the lower portion of the Linkage dialog box, highlight the line that contains the circuit node you want to link to the package by clicking the left mouse button at the line.
4. Make sure the package node of interest is on the active layer and zoom in so that you can select nodes in this specific area.
5. Move the cursor over to the package node of interest in the Layer View window and click on the node with the right mouse button. A flying menu (as shown) will appear.



Look here to confirm this is the package node of interest, then click “Link” to link the circuit node selected (in the Linkage dialog box) to the package node.

6. Confirm the package node name is the one that you want to link to, review the node information presented in the flying menu.
7. If the package node in the flying menu is the one you wanted, click Link. The selected package node name will appear in the highlighted line in the lower portion of the Linkage dialog box.



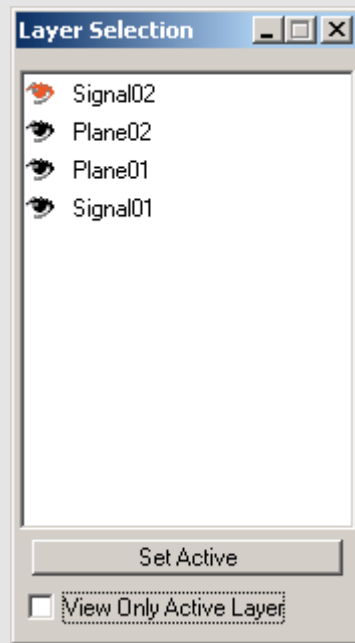
8. Click close to dismiss the Linkage dialog box.

► **To link several package nodes to a single circuit node:**

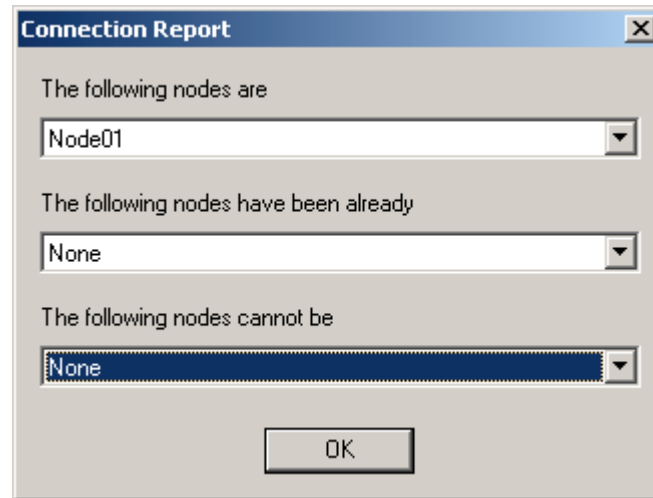
1. Open the Linkage dialog box by clicking Setup:Linkage from the menu bar. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. Select the package nodes of interest. If necessary, refer to [“To select certain kinds of objects:” on page 4-21](#).

**NOTE:**

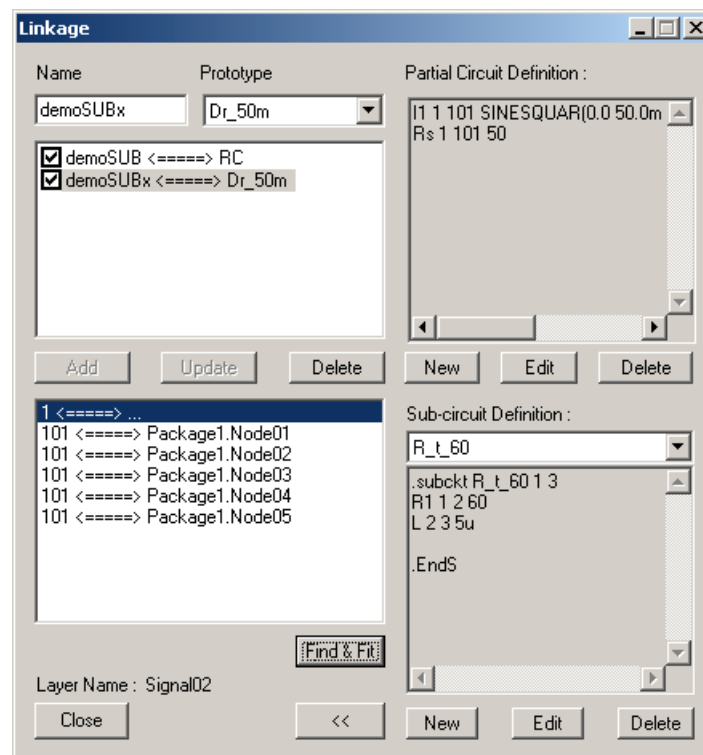
You may select nodes on several layers simultaneously by deselecting the View Only Active Layer option of the Layer Selection dialog box — as you select the nodes, hold down the Ctrl key. Or, use the Find function to select the nodes of interest — refer to [“Finding Objects” on page 4-113](#).



3. In the Linkage dialog box, highlight the line (in the upper circuit listing) that contains the circuit you want to select by clicking the left mouse button at the line.
4. In the lower portion of the Linkage dialog box, highlight the line that contains the circuit node you want to link to the package by clicking the left mouse button at the line.
5. Go to the menu bar and select Edit:Node:Link Selected Nodes. A Connection Report dialog box will appear. Review the report information, then click OK.



- The selected package node names will appear in the highlighted line in the lower portion of the Linkage dialog box. As shown in the example figure, circuit node 101 is connected at five package nodes.

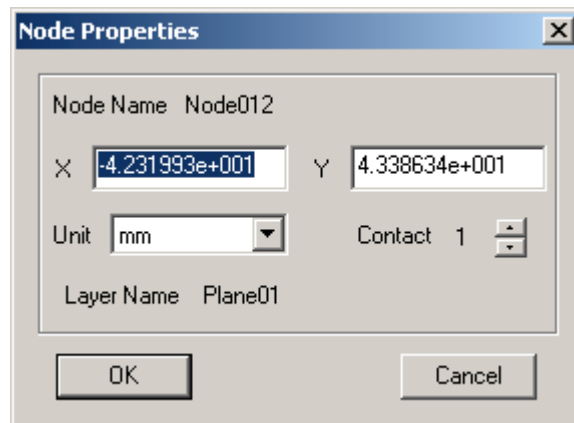


- Click close to dismiss the Linkage dialog box.

► **To link a circuit node to a shape:**

You can directly link circuit nodes to shapes (on planes or signal layers).

1. Activate the layer of interest. If necessary refer to [“Viewing Layers and the Layer Selection Dialog Box” on page 4-5](#).
2. Activate the View>Show Nodes menu option — so that you can see all the nodes on that layer.
3. Click on Add Node button from the Object toolbar.
4. Click on the location on the shape (plane or patch) where you want a package node, the Node Properties dialog box will appear.



5. Confirm the node makes contact with the shape — that is, Contact value equals 1. In fact, the default is always 1 if you so place a node onto a shape. The Contact value will be equal to 0 if you place an isolated node where there is no shape. For more information, refer to [“To alter node “contact” states:” on page 4-27](#).
6. Click OK.
7. Now, you may link the circuit node to the node on the patch. Refer to [“To make a linkage:” on page 5-9](#).

## Deleting a Linkage

If you make a mistake during the linking process, you can always delete the linkage.

### ► To delete a linkage:

1. Open the Linkage dialog box by clicking Setup:Linkage from the menu bar. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. In the lower portion of the Linkage dialog box, highlight the line that contains the node linking you want to select by clicking the left mouse button at the line.
3. Press the Delete key (on the keyboard) to remove the package node specification. The package node name will be removed and replaced by 3 dots (...).

### NOTE:

If the selected linkage is for a partial circuit node connected to more than one package node, the entire line will be removed.

## Modifying a Linkage

To modify a linkage, you have to delete the linkage first, and make another linkage. Refer to [“Deleting a Linkage” on page 5-14](#) then [“Linking Partial Circuits to the Package” on page 5-9](#).

## Checking Linkages

### ► To check linkages of circuit nodes:

1. Click Setup:Linkage from the menu bar to open the Linkage dialog box. Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) if necessary.
2. In the upper circuit listing of the Linkage dialog box, highlight the line that contains the circuit you want to select by clicking the left mouse button at the line.
3. In the lower linkage listing of the Linkage dialog box, highlight the line that contains the circuit node you want to check by clicking the left mouse button at the line. If the package node is on the current active layer, it will be marked as a green square in the Layer View window. If the package node is not on the current active layer, it will be marked as a gray square in the Layer View window. The name of the layer that the package node is on will be shown in the Layer Name field of the Linkage dialog box.



**NOTE:**

A **floating package node** is a package node connecting to only one object (either via or trace). That is, it is not connected to any partial circuit nodes nor more than one other package components (either via or trace). SPEED2000 will provide a warning notice of any floating package nodes, this message is for information only — the subsequent simulation is not affected.

## Selecting Circuits For Simulation

Use the check box, ☒ , that precedes each circuit in the Linkage dialog box to select and deselect circuits for simulation. This feature also permits you to override circuits deselected as a result of choices made in the Net Manager dialog box. Recall, by default, de-selecting a net causes all elements associated with the net to be de-selected for the subsequent simulation. For more information on the Net Manager, refer to [“Net Management” on page 4-15](#).

► **To select a circuit for participation in simulation:**

1. Click Setup:Linkage from the menu bar to display the Linkage dialog box. Observe the upper listing of the circuits in the package.
2. Place a check mark in the box that precedes the circuit of interest. You can toggle the presence or absence of the check mark by clicking on it.
3. Save the file to make the new information take effect for the next simulation.

Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) for an overview of the Linkage dialog box.

► **To deselect a circuit from participation in simulation:**

1. Click Setup:Linkage from the menu bar to display the Linkage dialog box. Observe the upper listing of the circuits in the package.
2. Remove the check mark in the box that precedes the circuit of interest. You can toggle the presence or absence of the check mark by clicking on it.
3. Save the file to make the new information take effect for the next simulation.

Refer to [“Overview Of The Linkage Dialog Box” on page 5-2](#) for an overview of the Linkage dialog box.



## Establishing Simulation Parameters

Before the .spd file (representing your package) is loaded into SPDSIM for simulation, simulation and viewing parameters need to be specified within the .spd file using SPDGEN. Although many simulation parameters have default values (see below), they may not be appropriate for your particular project. SPDGEN enables the User to conveniently change default simulation parameters. SPDGEN also provides several package component error checking functions to help ensure the integrity of the package information, refer to [“Checking Drawing Errors” on page 4-99](#). And finally, SPDGEN is used to specify the planes and the name of the Voltage Probability Distribution datafile. We discuss SPDGEN procedures to establish computational and viewing parameters in this chapter.

In addition to SPDGEN, which provides a convenient graphical interface to make changes to the .spd file, advanced users may want to use a text-editor to add and edit simulation parameter information directly in the .spd file. Refer to the [.spd File Format Reference](#) document (available on the Sigrity website: [www.sigrity.com](http://www.sigrity.com)) for more information on editing the .spd file using a text-editor.

### Default values for simulation and viewing parameters

**Mesh\_X = 60**

**Mesh\_Y = 60**

**Final time (equivalent to) Time steps = 100**

**View Step = sampling interval for data records (batch) = 10**

**Fmax (Transmission line) = 5 GHz**

**Plane Skin Effect = No**

**Enable Transmission Line Metal Loss = No**

**Dielectric Loss and Dispersion = No**

**Enable Initial DC Analysis = Yes**

**Passivity Check = No**

Simulation and viewing parameters are primarily established using the SetUp menu from the SPDGEN:Layer View window. Editing and adding package

components and linking circuits to the package is covered in previous chapters: [“Working with the Package” on page 4-1](#) and [“Working with Circuits” on page 5-1](#).

## Establishing Computation Parameters

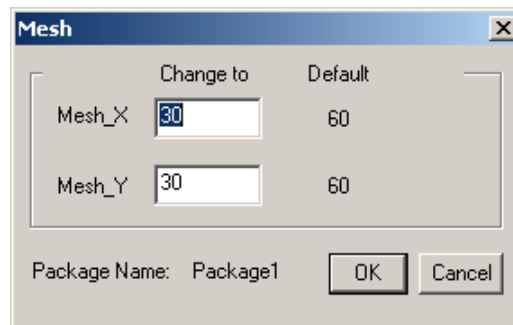
One of the most essential parameters is the mesh density. A uniform and rectangular mesh is used for the numerical discretization of voltages and currents between and on metal planes. Improper mesh setting may cause computational errors, and the SPDSIM module will halt processing of the .spd file if such errors occur. Refer to [“Checking Drawing Errors” on page 4-99](#), for more information.

The density of the mesh significantly affects the computation speed and memory requirement. So always try to choose the minimum mesh density required for the project. A special patented technology has been applied to the discretization process, so that a coarse mesh (as long as it has enough resolution to discretize the physical structure) will result in virtually the same computational accuracy as a fine mesh for most practical problems.

### *Establishing the numerical mesh*

#### **The Mesh Dialog Box:**

1. Click Setup: Mesh from the menu bar. The Mesh dialog box appears.
2. Type in the number of cells along the x and the y directions. For example, a value of 10 results in a 9x9 mesh cell matrix.
3. Click OK. To view the mesh refer to [“To turn on the mesh for viewing:” on page 6-3](#).



## NOTE:

We suggest when choosing mesh values to select values that will result in a mesh equal to or smaller than the minimum distance between vias, so that no adjacent vias are inadvertently discretized to a same mesh point.

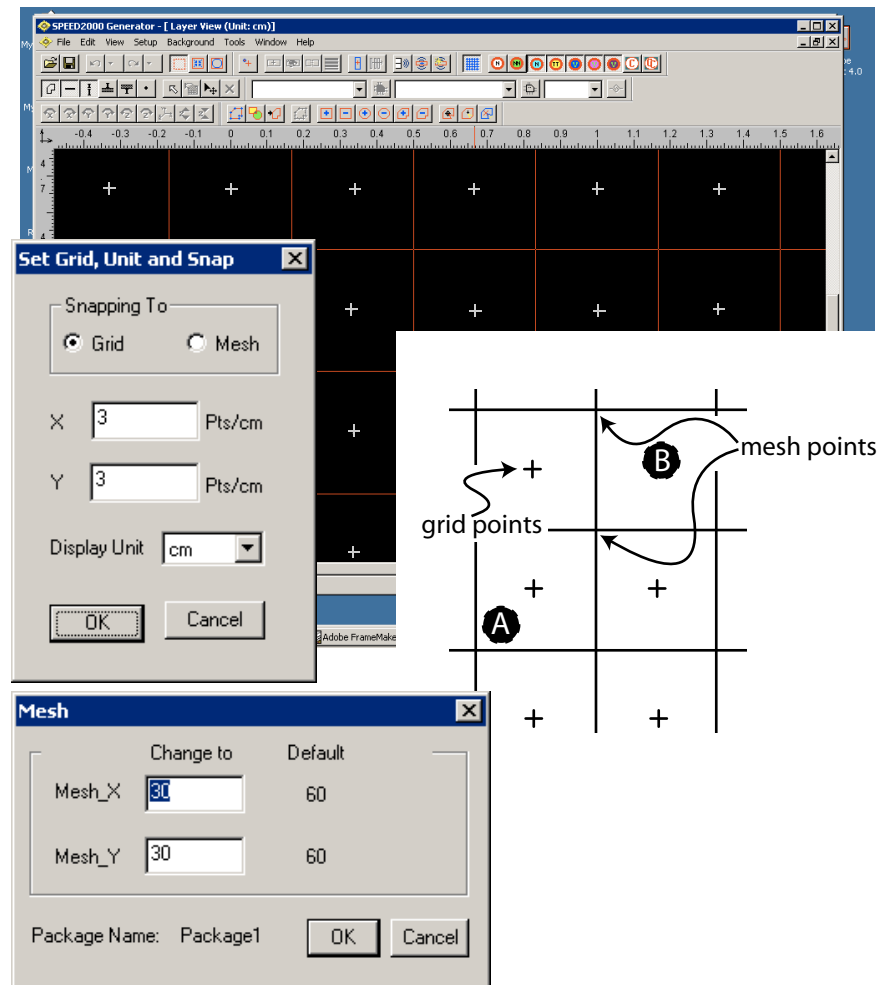


Mesh On/Off

## To turn on the mesh for viewing:

1. Click the Mesh On/Off button on the tool bar to view the mesh.

## Understanding The Mesh, The Grid and Snap to:



A “mesh point” is formed at the intersection of mesh lines specified using Setup:Mesh option. The mesh is specified using the Mesh dialog box (Click Setup:mesh) “Grid points” are specified by the user with the Set Grid, Unit

and Snap dialog box. The “snap to” drawing feature is turned on using the Set Grid, Unit and Snap dialog box. The default is snap to feature off.

**NOTE:**

Mesh Unit: a “mesh unit” is the distance between “mesh points”. For example, it is a good idea to set the length of a mesh unit equal to or smaller than the minimum distance between vias.

If “mesh snap to” is turned on, locations of package components, such as vias and end points of traces, are snapped to the nearest mesh points when they are added. If “grid snap to” is turned on, locations of package components, such as vias and end points of traces, are snapped to the nearest grid points when they are added. The location of components added prior to turning on the “snap to” function are not affected by turning on the snap to feature. Components moved after the snap to has been turned on will snap to the nearest mesh/grid point.

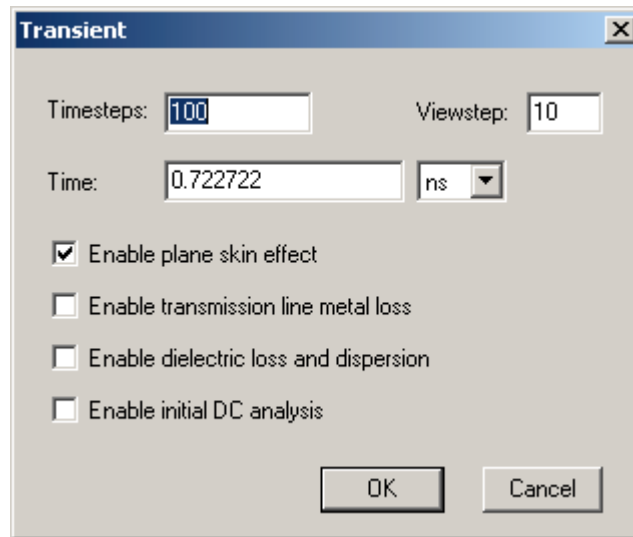
**Error Checking with the Mesh On**

Turning the Mesh On in conjunction with Error Checking causes the components to be numerically discretized to the nearest Mesh Point — however, this is just a virtual relocation of the components for the purpose of numerical computation and the components are not, in fact, relocated. After the mesh is turned on, the error checking functions search for component placement errors according to the virtually relocated physical layout information. Thus, a package structure with no actual drawing errors could be diagnosed as having errors (related to discretization) after the mesh is turned on.

## ***Establishing Transient Simulation Parameters***

### **The Transient Dialog Box**

1. Click Setup: Transient from the menu bar. The Transient dialog box appears.



### **Specifying Simulation Time**

There are two ways to specify the total simulation time. You can either type in the total time (shown in the box to the right of Time: ) or the number of time steps (shown in the box to the right of Timesteps:). These two values are correlated. Therefore when you type in or change the value of either one of the two fields, the other one will change accordingly. The unit of total simulation time can be selected between: s, ms, us, ns and ps.

#### **NOTE:**

The viewstep parameter establishes the sampling rate for display of 2D Curve information.

### **Enabling Metal Plane Skin Effect Option**

If the Enable Plane Skin Effect option is selected, the “frequency dependent” loss of the planes will be calculated during the transient simulation. By default only DC loss is considered for the planes.

#### **NOTE:**

Enabling plane skin effect will slow down the computation speed because convolution must be carried out.

**Enabling Transmission Line Metal Loss Option**

If the Enable Transmission Line Metal Loss option is selected, the “frequency dependent” loss as well as the DC loss of the traces (single or coupled transmission lines) will be calculated during the transient simulation. By default only lossless transmission line is considered.

**NOTE:**

Enabling transmission line skin effect will slow down the computation speed because convolution must be carried out.

**Enabling Dielectric Loss and Dispersion Option**

If the Enable Dielectric Loss and Dispersion option is selected, the frequency dependent loss and dispersion of dielectric will be calculated during the transient simulation. When this option is not selected, dielectric will be considered lossless (loss tangent will be ignored) and non-dispersive (if the dielectric constant is provided by a data file, the data at 1 GHz will be used).

**NOTE:**

Enabling dielectric loss and dispersion will slow down the computation speed because additional computer resources are needed to carry out the convolution.

**Enabling Initial DC Analysis Option**

If the Enable Initial DC Analysis Option is selected, the initial conditions of all the circuit elements will be calculated before the transient simulation starts. Planes and vias will be combined and considered as one node during the initial DC analysis if they belong to the same net. If you do not have initial values for C, L, V and I circuit elements, or if you do not have non-linear element, you can disable this option to reduce the overall computation time. .

**NOTE:**

The initial DC analysis is essentially a one-time-step transient simulation. If you want to calculate the distributed IR drop (DC drop) on planes, please refer to the file “Question and Answers” in QandA.PDF available on the Sig-  
rity website, [www.sigrity.com](http://www.sigrity.com) (see Customer Download area).

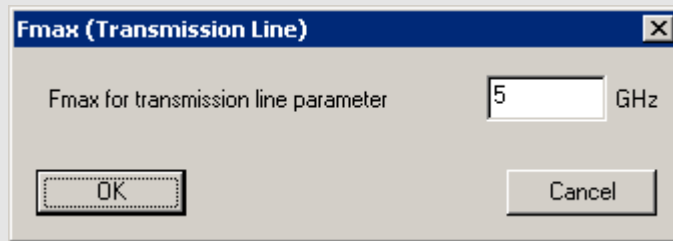
**NOTE:**

If you have a non-linear behavioral model in the circuit and you need to calculate an operation point, don't forget to enable the Initial DC Analysis option.



**NOTE:**

The highest frequency of interest in the simulation of lossy transmission line needs to be selected according to signal rise time or spectrum. It can be changed from Tools:Options: Fmax (transmission line). By default, the value is 5GHz.

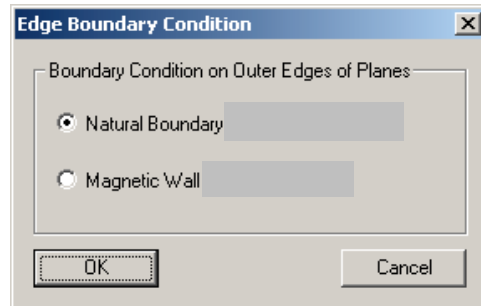
**Passivity Option**

It may occur that the time domain simulation does not converge if skin effect and/or the dielectric loss of the transmission lines is enabled. A more stable and slower scheme for the transmission line simulation will be used if the passivity checking option is enabled. The default is Passivity Check = No. To enable the Passivity Check select Tools: Options: Passivity Check.

## Edge Boundary Condition

The Edge Boundary Condition enables the User to simulate radiation loss at edges of metal planes. When the outer edges of metal planes are regarded as magnetic walls, it may effect simulation results above 1 or 2 GHz. Above that frequency, typically the resonant peaks will appear sharper than they should, due to the absence of radiation loss at plane edges.

The Natural Boundary Condition takes into account the frequency-dependent radiation loss at edges of metal planes and will help to bring the simulated results closer to reality. With the Natural Boundary condition selected, the peaks resulting in input impedance curves will generally be lower than those that will result with the Magnetic Wall boundary condition selected.



## NOTE:

The default is set to “Natural Boundary” while users have the option to select "Magnetic Wall" from the Edge Boundary Condition dialog box if, for example, comparisons with analytical solutions are needed (such as resonance frequencies).

### ► To set or reset the Edge Boundary Condition:

1. Select Setup:Edge Boundary Condition. The Edge Boundary Condition dialog box appears.
2. Select Natural Boundary or Magnetic Wall.
3. Click the OK button to record your selection and exit the Edge Boundary Condition dialog box.

## Choosing the Viewing Parameters

Dynamical variations of voltages and currents in package and circuits can be viewed during the simulation. Under the Setup:Simulation View menu from the menu bar, there are selections for viewing:

- circuit voltage (CKT Voltage),
- package voltage (PKG Voltage),
- current (Current),
- 3D spatial voltage (3D).

## ESTABLISHING SIMULATION PARAMETERS

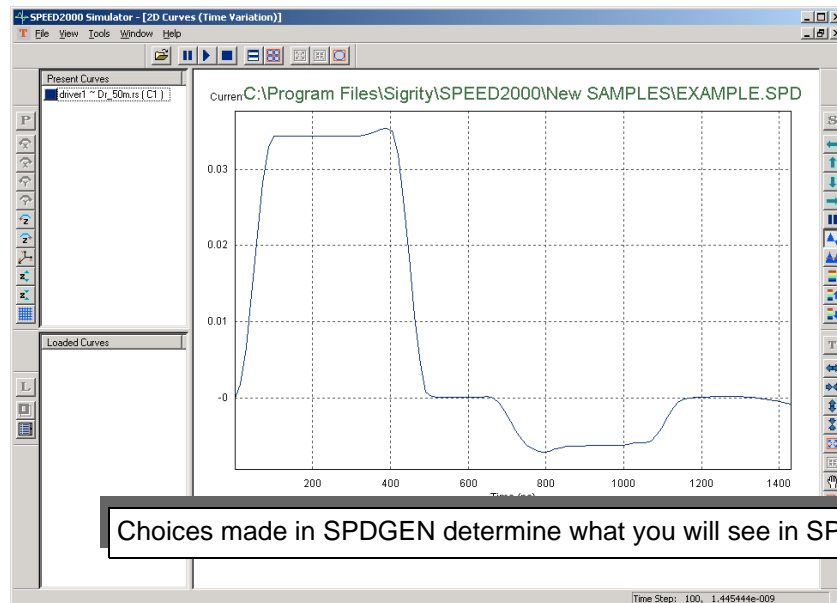
Each of these selections will open a dialog box for specifying the corresponding parameters. In each of these dialog boxes, there is a Line Control field which is common to all these 4 selections.



The New button in the Line Control field enables you to create a new entry, the < and > buttons are for you to go backward and forward in the entry list, and the Delete button is for clearing out the current entry.

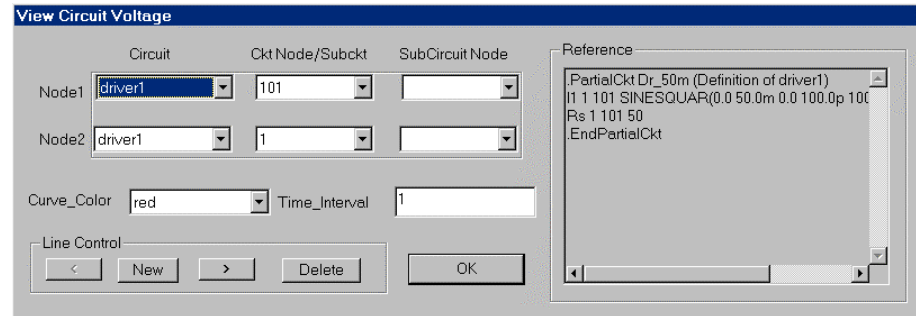
### ***Choosing Circuit Voltages for Viewing***

The View Circuit Voltage dialog box enables the User to specify that SPDSIM calculate and display the potential difference between any two circuit nodes during the simulation. The results (shown below) will be displayed dynamically as a 2D curve during the simulation.



**The View Circuit Voltage Dialog Box:**

1. Click Setup: Simulation View: CKT Voltage from the menu bar. The View Circuit Voltage dialog box appears.



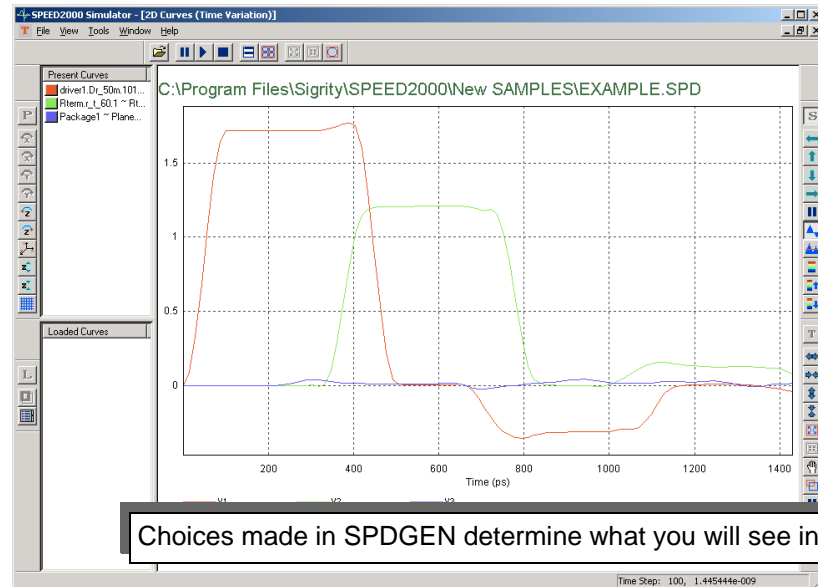
2. If there is an entry displayed in the View Circuit Voltage dialog box, click on the > button to get to the entry you want to modify, or click the New button to get a blank entry for a new curve.
3. For the fields of Node1, first select the circuit in the Circuit list box, then select the circuit node in the Ckt Node/Subckt lists box. If the circuit node is in a sub-circuit, after selecting the circuit, you need to select the subcircuit in the Ckt Node/Subckt list box, then select the subcircuit node in the SubCircuit Node list box.
4. Selections of fields for Node2 can follow the same procedure as above.
5. Select the color for the curve in the Curve\_Color list box.
6. Specify the time interval between output results. This option will make the simulator draw a point every n time steps, where n is the number specified as the time interval field.
7. Click the New button to add more curves for observation, or click OK to exit the View Circuit Voltage dialog box.

**NOTE:**

After a circuit is selected, the definition that the circuit uses will appear in the right-hand side window in the View Circuit Voltage dialog box.

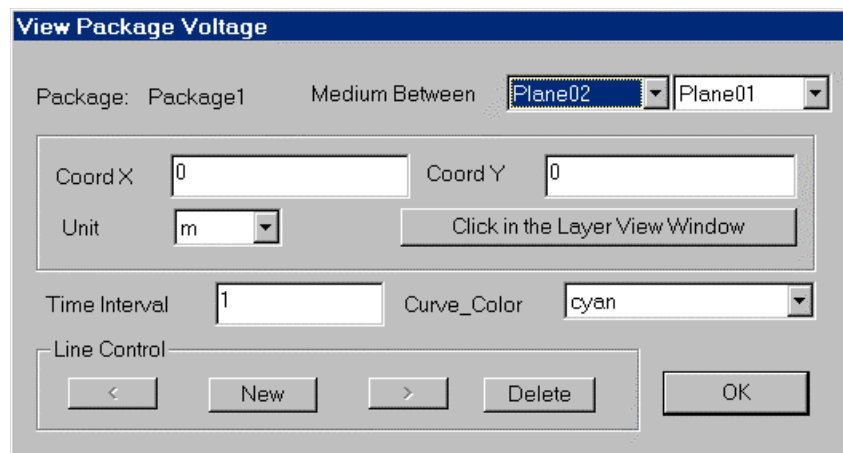
## Choosing Package Voltages for Viewing

The View Package Voltage dialog box enables the User to specify that SPDSIM calculate and display the potential difference between any adjacent planes during the simulation. The results (shown below) will be displayed dynamically as a 2D curve during the simulation.



### The View Package Voltage Dialog Box:

1. Click Setup: Simulation View: PKG Voltage from the menu bar. The View Package Voltage dialog box appears.



2. If there is an entry displayed in the View Package Voltage dialog box, click on the > button to get to the entry you want to modify, or click the New button to get a blank entry for a new curve.

3. Select the pair of adjacent metal planes in one of the two boxes to the right of Medium Between. The left box is the name of the upper metal plane while the right box is the name of the lower metal plane. Once you select one metal plane in one of the two boxes, the other plane is determined automatically.
4. Type in the x and y coordinates.
5. Select the proper unit for the coordinate in the Unit list box.

**NOTE:**

Another way to get the x and y coordinates of the observation point is to click on the Click In The Layer View Window button in the View Package Voltage dialog box, then use mouse click in the Layer View window to fetch the desired location.

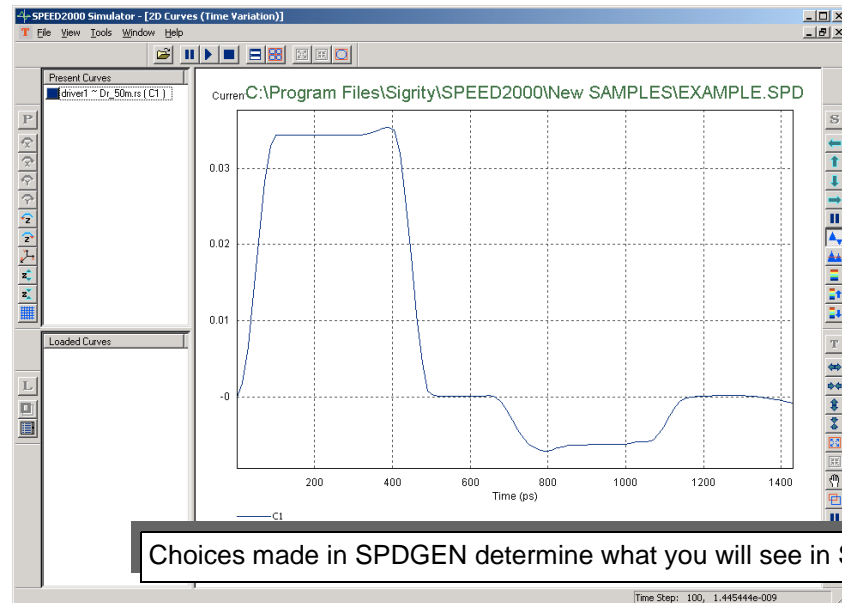
6. Select the color for the curve.
7. Specify the Time Interval between output results. This option will make the simulator draw a point every n time steps, where n is the number specified in the time interval field.
8. Click the New button to add more curves for observation, or click OK to exit the View Package Voltage dialog box.

**NOTE:**

When you open the View Package Voltage dialog box, if there is an entry already specified, the observation location would be marked with a yellow symbol  $\oplus$  in the Layer View window.

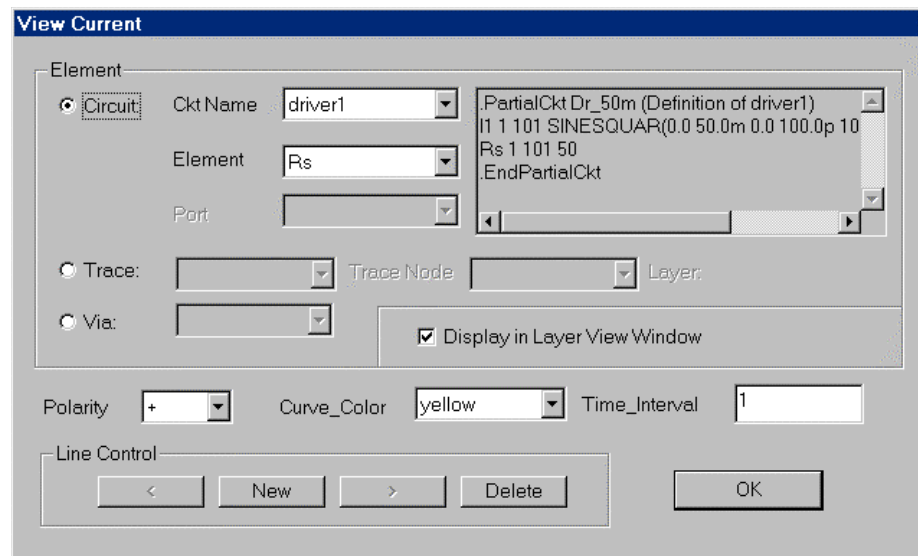
## Choosing Circuit/Package Currents for Viewing

Currents in circuit elements (R, L, C, T, V and I) and in package objects (vias and traces) can be viewed dynamically during the simulation (in SPDSIM) as 2D curves (shown below). Use the View Current Dialog box to select package components for viewing.



### The View Current Dialog Box (Circuit):

1. Click Setup: Simulation View: Current from the menu bar. The View Current dialog box appears.

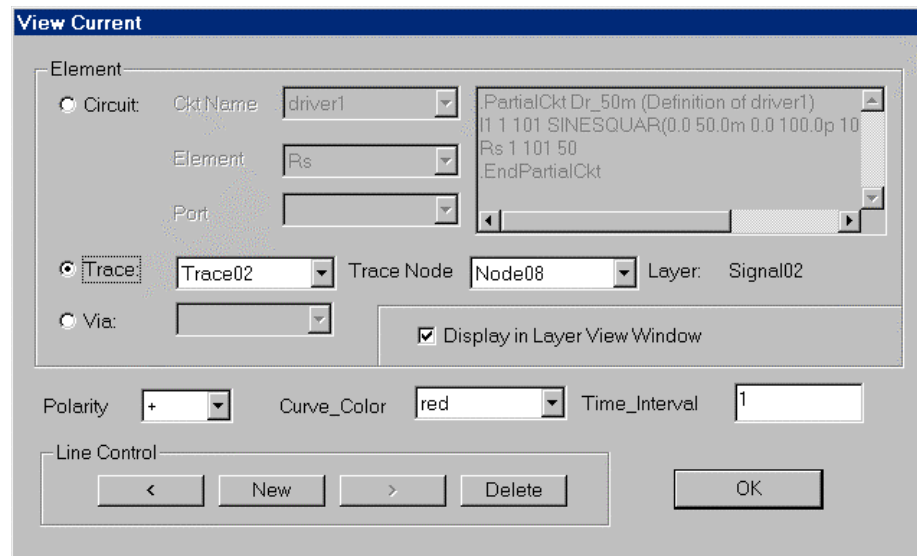


2. If there is an entry displayed in the View Current dialog box, click on the > button to get to the entry you want to modify, or click the New button to get a blank entry for a new curve.
3. Click in the circle right next to the Circuit option in the View Current dialog box as shown in the above figure.
4. Select the circuit in Ckt Name list box. The definition of the circuit will appear in the window in the upper right corner of the View Current dialog box.
5. Select the circuit element in the Element list box.
6. Choose the direction of the current flow in the Polarity list box. + sign means current flows from the positive node toward the negative node. - sign means the opposite direction.
7. Select the color for the curve in the Curve\_Color list box.
8. Specify the time interval between output results. This option will make the simulator draw a point every n time steps, where n is the number specified in the time interval field.
9. Click the New button to add more curves for observation, or click OK to exit the View Current dialog box.



**The View Current Dialog Box (Trace):**

1. Select the trace in the Layer View window.
2. Click the *Observe* button on the tool bar, the View Current dialog box will appear. The name of the selected trace is shown in the Trace Name list box. The layer that contains the trace is shown in the Layer field.



3. Select one of the two ends of the trace in the Trace Name list box. The selected node will be marked as a yellow square in the Layer View window.
4. Choose the direction of the current flow in the Polarity list box. + sign means current flows from the positive node toward the negative node. - sign means the opposite direction.



5. Select the color for the curve in the Curve\_Color list box.
6. Specify the time interval between output results. This option will make the simulator draw a point every n time steps, where n is the number specified in the time interval field.
7. Click the New button to add more curves for observation, or click OK to exit the View Current dialog box.

**NOTE:**

When the entry in the View Current dialog box is for trace, you will see the selected end of the trace is marked as a yellow dot (on active layer) or a gray dot (on inactive layer) in the Layer View window.

**The View Current Dialog Box (Via):**

1. Select the via in the Layer View window.
2. Click the *Via Property* button on the tool bar to open the Via Property dialog box.
3. Select the section of the via and click the *Observe* button in the Via Property dialog box. The View Current dialog box appears. The name of the selected via is shown in the Via Name list box.

4. Choose the direction of the current flow in the Polarity list box. + sign means current flows from the positive node toward the negative node. - sign means the opposite direction.
5. Select the color for the curve in the Curve\_Color list box.
6. Specify the time interval between output results. This option will make the simulator draw a point every n time steps, where n is the number specified in the time interval field.

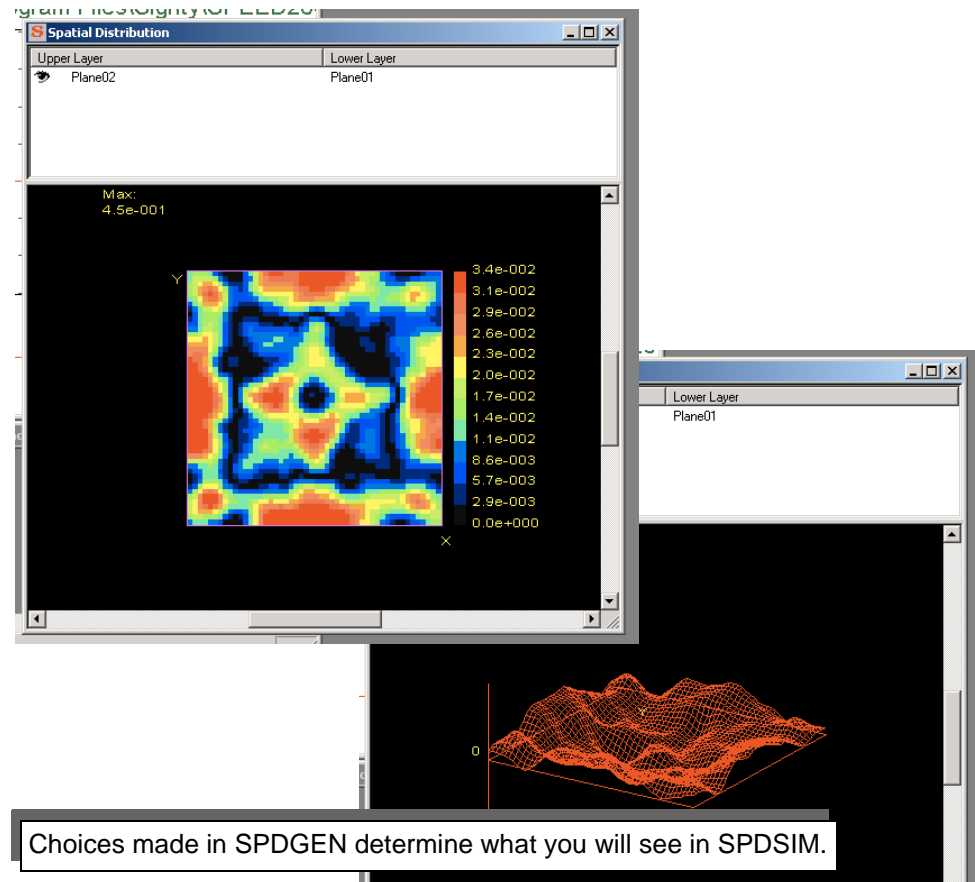
7. Click the New button to add more curves for observation, or click OK to exit the View Current dialog box.

## NOTE:

When the entry in the View Current dialog box is for via, you will see the selected via is marked as a yellow dot (on active layer) or a gray dot (on inactive layer) in the Layer View window.

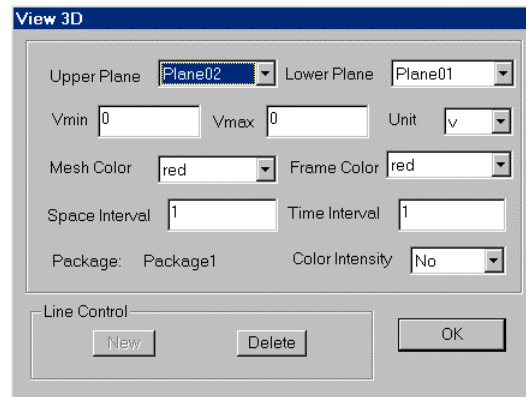
## Choosing 3D Spatial Voltages for Viewing

The View 3D dialog box enables the User to specify that SPDSIM calculate and display the spatial voltage distribution between any adjacent planes during the simulation. The results (example shown below) will be displayed dynamically as a wireframe or color intensity plot, within the Spatial Distribution window of SPDSIM, during the simulation. Note the Upper and Lower Plane specification panel. The spatial voltages for the adjacent planes selected (here in SPDGEN) will be the one shown on screen when simulation starts. However, the derived spatial voltages between other (non-adjacent) layers will also be available for viewing after the initial simulation run.



**The View 3D Dialog Box:**

1. Click Setup:Simulation View:3D, the View 3D dialog box appears.



2. Click New to start a new entry if nothing has been selected. Select the pair of adjacent metal planes in Upper Plane or Lower Plane list box. Once one of the two planes is selected, the other plane is determined automatically.
3. Select the color for the surface plot in the Mesh Color list box.
4. Select the color for the zero volt reference frame in the Frame Color list box.
5. Specify the Space Interval between output results. This option will make the simulator draw results every m mesh points in the x and y directions, where m is the number specified in the Space Interval field.

**NOTE:**

Displaying of 3D spatial voltages can take considerable amount of time, especially for dense meshes, during simulation. By choosing an integer other than 1 in the Space Interval field, the spatial voltage will be displayed with a coarser sampling points but a faster animation speed.

6. Specify the Time Interval between output results. This option will make the simulator draw results every n time steps, where n is the number specified as in the Time Interval field.
7. In the Color Intensity box, set the initial drawing to be the surface plot or the color intensity plot. In SPDSIM, you can switch between the surface plot and the color intensity plot.
8. Type in the values for Vmin and Vmax and select their corresponding unit...

**NOTE:**

Leaving both Vmin and Vmax to zero will make the 3D animation use auto scale.

9. Click OK

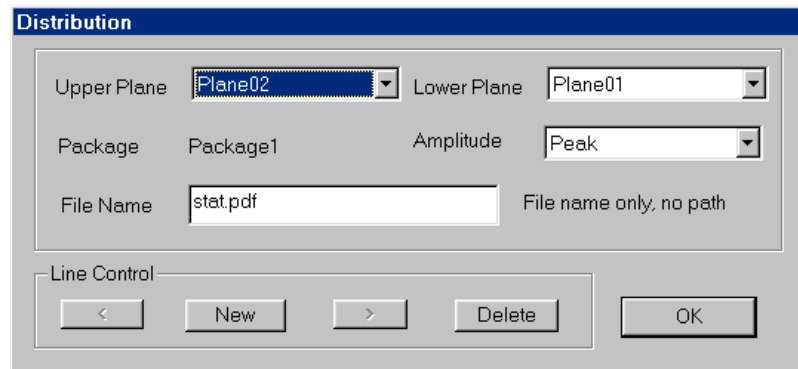
### ***Choosing Parameters for Voltage Viewing and Analysis***

A spatial animation of peak or average voltage values may be viewed during the simulation (refer to [“Spatial Distribution Window” on page 3-33](#)) and (if selected) statistical calculations are applied to the simulation data to derive the distribution of voltage amplitude — this data is plotted to create the Probability Distribution curve (refer to [“2D Curves Window” on page 3-13](#)).

Either the peak voltage value or the average voltage value can be used as the indicator. Peak voltage indicates the maximum voltage swing that each location experienced during the time period of the simulation. Average voltage records the average amplitude of the voltage at each location during the simulation. Transient voltage values are sorted to form the probability distribution curve available in SPDSIM. The Probability Distribution curve can be used to analyze the noise voltage distribution of the package and evaluate ground/power bounces.

#### ***The Distribution Dialog Box:***

1. Click Setup:Distribution from the menu bar, the Distribution dialog box appears.



2. Select the pair of adjacent planes in the Upper Plane or Lower Plane list box. Once one of the two planes is selected, the other plane is determined automatically.
3. Select the voltage amplitude to be either Peak or Average. Peak voltage indicates the maximum voltage swing that each location experienced during the time period of the simulation. Average voltage records the average amplitude of the voltage at each location during the simulation.
4. Type in the name of the output file. Do not include any path before the file name since the file will always be generated in the same folder of the SPD file. The file can have any extension, the default is .out.
5. Click OK

## Far Field (Edge) Radiation

Radiation calculation in SPEED2000 is a simple two-step operation: you first need to identify in SPDGEN the traces/edges that are to be considered for radiation calculation and then, in SPDSIM, after obtaining voltages and currents through the transient simulation, you need to specify the measurement parameters such as the antenna location, signal period and observation range.

For more information on the radiation calculation options which complement our SPEED2000 product and the setup in SPDGEN that is necessary, refer to [“Calculating Radiation” on page 8-1](#) for more information.

## ESTABLISHING SIMULATION PARAMETERS

## Simulation and Post Processing

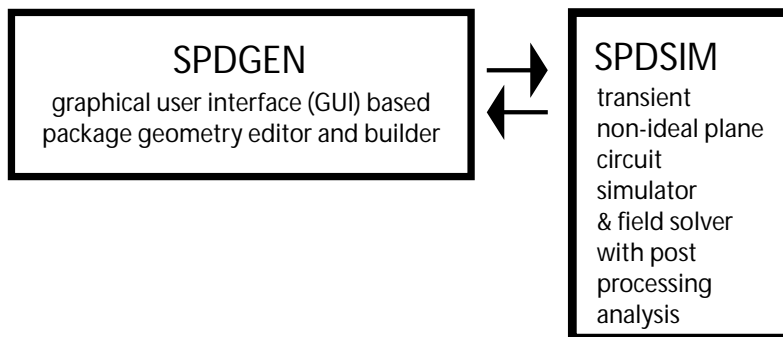
SPEED2000 enables package and multi-chip module simulation to assist the engineer with electrical evaluation and design. It provides a fast and accurate dynamic, full wave electromagnetic analysis and simulation that takes into account all electromagnetic interactions between power and ground planes and traces, and all via-to-via coupling effects. The results are displayed as 2D and 3D animations on the User's PC and complemented with post processing data analysis tools. SPEED2000 is composed of two executable modules — SPDGEN and SPDSIM. SPDGEN provides a graphical user interface so the user can conveniently build and modify package components and establish simulation parameters. SPDSIM provides the transient simulation engine that processes and displays the package components — SPDSIM also provides a post processing interface so the user can analyze, compare and print the results of package simulation.

This chapter details the most commonly used simulation and post processing procedures. For a comprehensive discussion of all of the features, menus, toolbars, buttons and dialog boxes of SPDSIM, refer to [“SPDSIM Features Overview” on page 3-1](#). For more information on SPDGEN, refer to [“SPDGEN Features Overview” on page 2-1](#). SPEED2000 also provides a far field radiation analysis function, refer to [“Calculating Radiation” on page 8-1](#).

### SPEED2000 Data File Flow

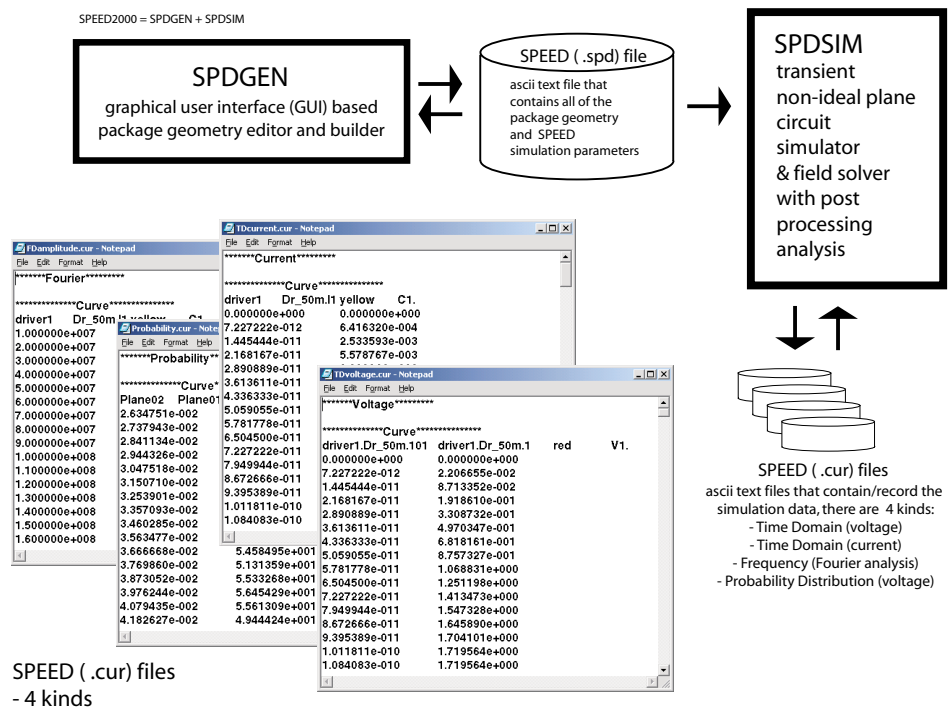
The data of SPEED2000 is maintained in unique (text) file formats. We discuss the SPEED2000 data flow in this section. SPEED2000 is composed of two executable modules, SPDGEN and SPDSIM. The SPDGEN module provides

SPEED2000 = SPDGEN + SPDSIM



the GUI-based package geometry editor and file builder, it is used to create and

edit the SPEED file (.spd) which contains package geometry and simulation parameters. The SPDSIM module loads and executes a transient simulation of the SPEED file (.spd) created in SPDGEN — it should be noted that SPDSIM does not change the SPEED file contents. SPDGEN’s Graphical User Interface (GUI) makes convenient “what if” analysis of a package design possible; that is, the user can conveniently engineer a possible solution to a problem (made apparent during the simulation) and then simulate the new model to see the results of changes made. It should also be noted that selecting which components (of a package design) will actually be simulated is performed using SPDGEN. For more information refer to [“Establishing Computation Parameters” on page 6-2](#).



### SPEED2000 Data File FLOW

The output file from the SPDGEN module is the SPEED file (.spd). The SPDSIM module processes the SPEED file information and performs a transient simulation of the package components and circuits selected for simulation. The user can not edit or save the SPEED file using SPDSIM. Transient simulation data is displayed as 2D and 3D animations within SPDSIM and the data generated during the simulation is saved in SPEED Curve files (.cur). There are four kinds of SPEED Curve files that may be saved from a SPEED2000 simulation: Time Domain (current and voltage information), Frequency Domain (a Fourier analysis series) and Probability Distribution of voltage values between layers. Saved Curve files can be reloaded into SPDSIM for viewing, comparison and



printing. The .spd and .cur files are both ascii format files. Their contents can be viewed and edited using a text editor. The SPEED file format (.spd) is described in detail in the [.spd File Format Reference](#) document (available on our website). The SPDGEN module is described [“SPDGEN Features Overview” on page 2-1](#).


#### NOTE:

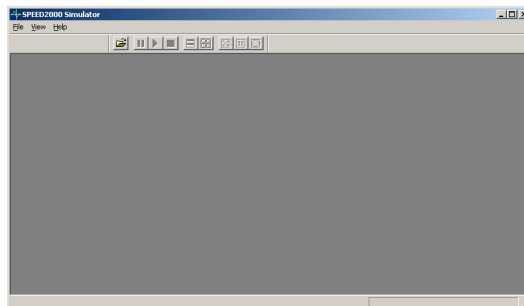
If the viewing and simulation parameters have been set to produce it, there is another intermediate data file produced related to the probability distribution feature (xxxxxstat.out). For more information refer to [“Choosing Parameters for Voltage Viewing and Analysis” on page 6-18](#)

In addition, the data for spatial distribution of voltage between metal planes and spatial distribution of parallel-mode currents on plane surfaces (UP and Down) can be stored in data files (as it is generated). At this time it requires that the User manually insert statements to the .spd file (using a text editor). Please refer to the [.spd File Format Reference](#) document for additional information — see the Output3Dvoltage and .Output3DCurrent sections.

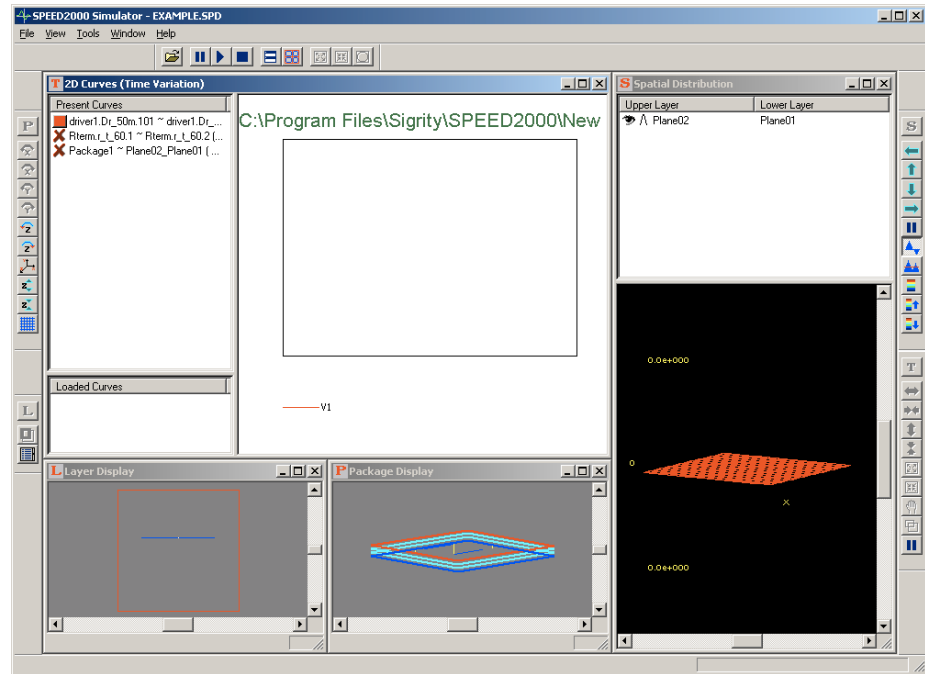
## Starting Simulation



Launch the SPDSIM executable module from the Programs menu of Windows or double click the SPDSIM icon  in the SPEED2000 folder. After starting SPDSIM, click File:Open from the menu bar to load a SPEED2000 native format file (.spd). If the .spd file is incomplete, or it has errors detectable by SPDSIM, it will fail to open the file and prompt you to check the error log of the problematic .spd file. The error log file produced will have the same file name as the SPD file, but with the xxxxxxxx\_SPDSIM.err suffix. Where xxxxxx equals the input .spd file name.



After reading in the .spd file, the package component information will be shown in both 3D Package Display and 2D Layer Display windows.



After the SPD file is opened, four windows will appear:

- The 2D Curves (Time Variation) window displays 2D curves representing voltages and/or currents versus time.
- The Layer Display windows shows the top down view of each layer components.
- The Package Display window shows the package components in a dimensional display that is rotatable about the x, y and z axis.
- The Spatial Distribution window displays spatial variation of voltages between planes in wireframe animation or color intensity plot mode.

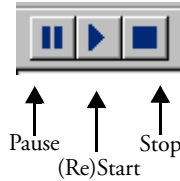
For comprehensive details of the features, toolbars, buttons, dialog boxes and menu options of SPDSIM, refer to: [“SPDSIM Features Overview” on page 3-1](#)

#### NOTE:

If the current window display mode is set to 8-bit (256) color, the display of 2D curves and Spatial Distribution will take much less time, when compared to 16-bit or 24-bit color mode. Refer to Microsoft Windows Operating System > Display Properties > Settings tab for more information.

## Controlling Simulation

To start the simulation, click the start button on the SPDSIM tool bar. The time step shown at the lower right corner of the main program window starts ticking as simulation proceeds.



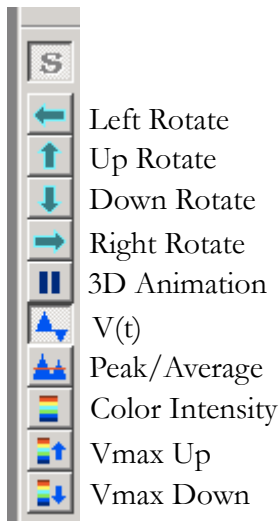
After the simulation starts, you can pause the simulation by clicking the Pause button. Clicking the Pause button again will let you continue the simulation. You can also stop the simulation at any time by clicking the Stop button. The (Re)start button will permit you restart the simulation from the beginning.

To speed up the simulation, you can pause the animation in the Spatial Distribution window and/or the animation in the 2D Curves window. The Pause button in the “S” tool bar will pause the 3D animation and increase the simulation speed by about 600%. The Pause button in the “T” tool bar will pause the 2D animation and increase the simulation speed by 50%~200%. To continue the animation, you just click the Pause button again.

For comprehensive details of the features, toolbars, dialog boxes and menu options of SPDSIM, refer to: [“SPDSIM Features Overview” on page 3-1.](#)

## The Spatial Distribution Window

The Spatial Distribution Window permits the user to view the transient simulation in wireframe animation or color intensity plot views. The Spatial Distribution window has two panels — the upper panel enables the user to control which layer pair simulations will be displayed in the lower transient simulation panel. Refer to [“Spatial Distribution Window” on page 3-33](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of the SPD-SIM:Spatial Distribution window. Following are some introductory instructions:



- The user controls which layer pairs will be visible during the transient simulation by clicking on the layer pair list. Click once to enable the display (an open eye appears). Click again to hide the layer pair (the eye closes).
- During and after the simulation, the Spatial Distribution window can be controlled through the “S” toolbar to show the spatial distribution of:
  - transient voltage fluctuations between any pair of planes
  - or, peak/average voltage fluctuations between planes.
- When the V(t) button is pressed, the Spatial Distribution window shows the dynamic voltage fluctuations, while clicking on the Peak/Average button will make the window show the peak/average voltage distribution. Recall, transient simulation viewing and calculation parameters are established using the SPDGEN module. Refer to [“Establishing Simulation Parameters” on page 6-1](#) for more information.

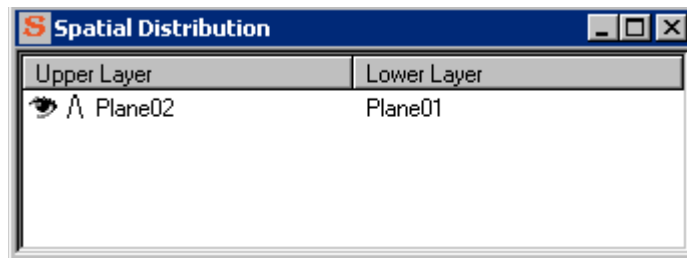
- The Pause button can pause and resume the drawing during the simulation. The simulation proceeds faster if the wireframe animation or color intensity plot in the Spatial Distribution window is paused.
- The Rotation buttons can be used to adjust the viewing angle of the wire frame plot.

**NOTE:**

Refer to [“SPDSIM Features Overview” on page 3-1](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of SPD-SIM.

**To view the spatial distribution of voltage between adjacent layer(s):**

1. Activate the Spatial Distribution window.



2. Click on the upper plane name to toggle between on/off mode. Notice the eye icon will toggle open/close to indicate on/off during your selection.
3. Continue to select the plane pairs for viewing.
4. Readjust the Spatial Distribution window size.
5. Activate the lower panel of the Spatial Distribution window and click the Fit button on the main tool bar to get the best view.

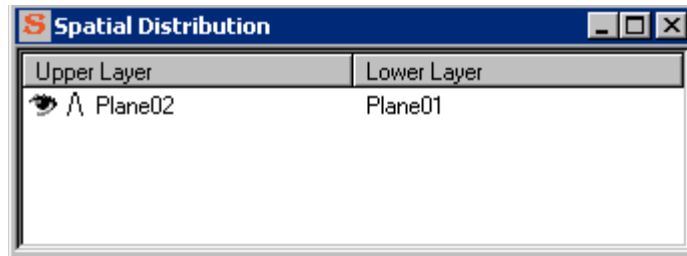
**NOTE:**

The main toolbar buttons for Zoom In, Zoom Out and Fit are only available when the lower panel of the Spatial Distribution window is active. Click in the lower panel to make it active.

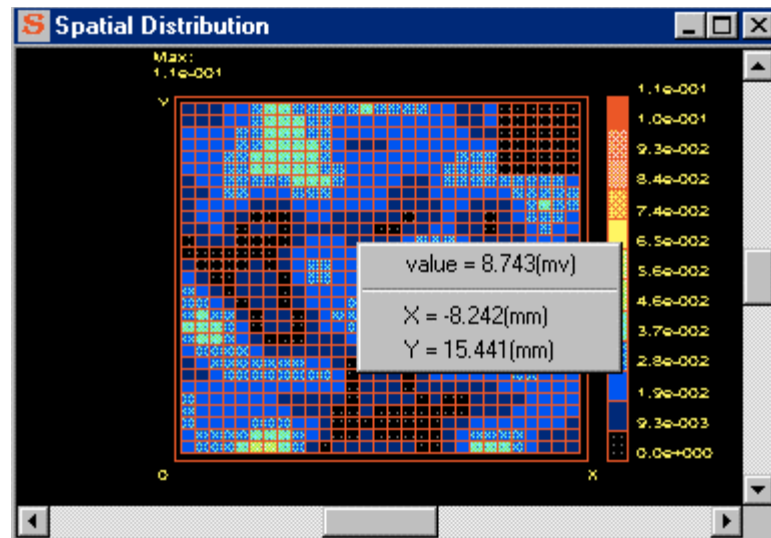
**To view the voltage value at a selected location:**

When the Spatial Distribution window is in the color intensity mode, the simulated voltage value at any location can be displayed.

1. Activate the Spatial Distribution window.



2. Click on the Color Intensity mode button.
3. Click on the upper layer plane name to toggle between on/off mode. Notice the eye icon will toggle open/close to indicate on/off during your selection.
4. Continue to select the plane pairs for viewing.
5. Readjust the Spatial Distribution window size so all of the layers selected for viewing can be seen.
6. Run the simulation. Click on the (Re)Start button. Click the pause button when the simulation displays an event of interest.
7. Click the right mouse button at any location in the color intensity plot. A flying menu will appear showing the voltage value as well as the coordinates of the location.



## The 2D Curves Window

The SPDSIM 2D Curves window displays the simulation data (in 2D curve plots) that results from the SPDSIM transient simulation of package components. The 2D Curves window is the user interface to access post processing tools for data analysis and viewing — analysis tools are: Time Domain display of voltage and current simulation data, Frequency (Fourier series) display of simulation data and Probability plot of voltage over simulation time. The 2D Curves window can be used for:

- viewing voltages and currents in both time domain and frequency domain;
- viewing the computed results, such as impedance and transfer functions, in frequency domain;
- viewing the statistical distribution function of the power/ground noise voltages.
- viewing a 2D animation of the potential difference between any two circuit nodes or any adjacent package planes. Viewing Circuit/Package Voltages

Refer to [“2D Curves Window” on page 3-13](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of the SPDSIM:2D Curve Display window.

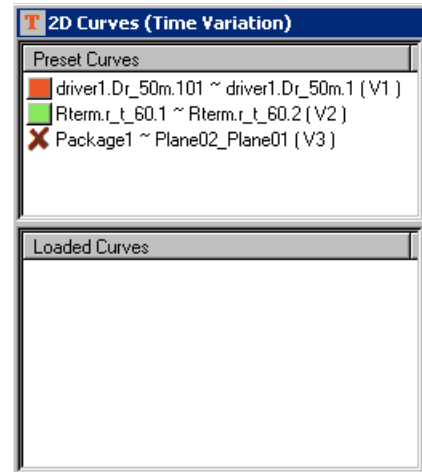
As soon as the simulation starts, the voltages and currents, which have been specified for observation using SPDGEN, are available to be displayed in the 2D Curves window.

### To view circuit/package voltages:

1. Activate the 2D Curves window by clicking anywhere in the window.
2. Select Tools:Time Domain:Voltage from the menu bar. The title bar of the 2D Curves window says 2D Curves (Time Variation).
3. Select the curves for viewing in the Present Curves panel. Refer to [“Choosing the Viewing Parameters” on page 6-8](#) for more information.
4. Click the (Re)Start button. The data for the curves selected for viewing is plotted as it is calculated during the simulation.

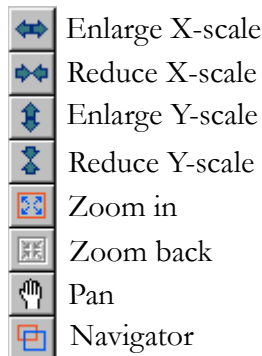
### Viewing Transient Circuit/Package Currents

As soon as the simulation starts, the currents, which have been specified for observation using SPDGEN, are available to be displayed in the 2D Curves window. (Refer to [“Choosing the Viewing Parameters” on page 6-8.](#) )



#### To view circuit/package currents:

1. Activate the 2D Curves window by clicking anywhere in the window.
2. Select Tools:Time Domain:Current from the menu bar. The title bar of the 2D Curves window says 2D Curves (Time Variation).
3. Select the curves for viewing in the Present Curves panel. Refer to [“Choosing the Viewing Parameters” on page 6-8](#) for more information.
  - Click the (Re)Start button. The data for the curves selected for viewing is plotted as it is calculated during the simulation. The first four buttons on the “T” tool bar are for controlling the viewing scale of the X-axis and Y-axis in the 2D Curves window. The last four buttons are related to displaying the details of the curves. Their functions are depicted in the diagram on the left.



#### NOTE:

Refer to [“SPDSIM Features Overview” on page 3-1](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of SPD-SIM.

#### To export the curve display window as a bitmap file:

1. Choose File->Export->Image (Bitmap File). The Save As dialog box is displayed.
2. Enter the name of the file into the *File name:* field of the Save As dialog box and press save. The current curve display window is saved as a bitmap file ready to be inserted into a report.

## The Layer Display Window

The SPDSIM:Layer Display window provides a convenient top down view of the package components selectable on a per layer basis as traces and/or shapes. Refer to [“Layer Display Window” on page 3-39](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of the SPDSIM:Layer Display window.

## The Package Display Window

The SPDSIM:Package Display window provides a convenient dimensional view of the package components — rotatable on the X, Y, Z axis. Refer to [“Package Display Window” on page 3-42](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of the SPDSIM:Package Display window.

## Post Processing

SPDSIM models circuits based on Fourier Theory during a transient simulation of package information contained within the SPEED (.spd) file. Simulation data is available for 2D plots and post processing analysis using the Tools menu. The following figure summarizes the calculations made:

<u>Fourier Theory Based Calculations/Plots</u>	
<u>Time Domain</u>	Voltage over time
	Current over time
<u>Frequency Domain</u>	Amplitude over frequency
	Phase Shift over frequency
	Real Part over frequency
	Imaginary Part over frequency
<u>Statistical Plot (Voltage Probability)</u>	

When a simulation is finished (at the end of the specified simulation time) or stopped (if the Stop button on the tool bar is clicked), simulation data can then be processed, viewed, compared and analyzed for various purposes.

### **Viewing Probability Distribution of Power/Ground Plane Noise**

If you have specified the statistical distribution parameters in SPDGEN (refer to [“Choosing Parameters for Voltage Viewing and Analysis” on page 6-18](#)),



then either the peak or the average voltage distribution data between planes is saved during simulation (as the xxxxxnamedStatistics.out file). The simulated voltage data can be sorted out and plotted as a voltage probability distribution function — to quantify the severity of the power/ground plane noise in the model. The data can be saved and compared to future simulations of the model after changes have been made.

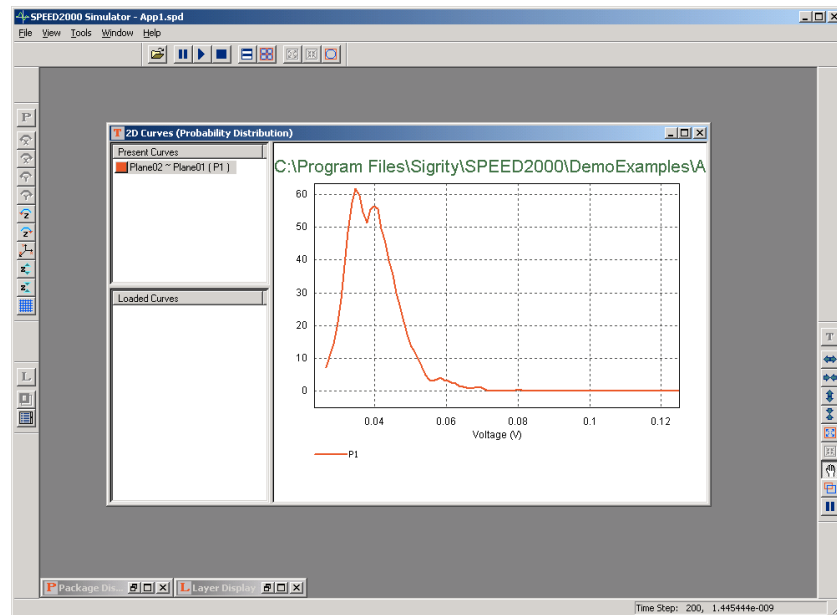
**To view the probability distribution function of the noise voltage:**

1. Activate the 2D Curves window by clicking anywhere in the window.

**NOTE:**

The Probability Distribution of the discrete random variable (voltage) is plotted by a function which displays the probability (0 to 100%) that the voltage equals a specific value between specified planes, for each value of voltage in the simulated range. Two parameters can be adjusted with respect to the display of the Probability Distribution plot: nBars and Factor. Refer to [“Tools Menu” on page 3-17](#) for details of the Preferences dialog box

2. Click Tools:Probability from the menu bar. The probability distribution curve/curves should appear in the 2D Curves window. Recall, you must specify the statistical distribution parameters using the SPDGEN module. For detailed information refer to [“Choosing Parameters for Voltage Viewing and Analysis” on page 6-18](#).



3. For example, the probability distribution curve above shows that the noise voltage spreads from .026V to .054V, and for 50-60% of the area between

the planes, the noise voltage is in the range of .02V to .045V. This type of statistical distribution curves may be used to compare and evaluate the effects of de coupling capacitors and different design modifications on package power/ground supply systems.

### **Generating Fourier Transforms of Transient Data**

Through Fourier transforms, the time domain voltages and currents can be viewed in the frequency domains of:

- Amplitude
- Phase Shift
- Real Part
- Imaginary Part

#### **To view voltages in the frequency domain:**

1. Activate the 2D Curves window by clicking anywhere in the window.
2. Check Tools: Time Domain: Voltage from the menu bar.
3. Select those curves which will be transformed into frequency domain using the Present Curves panel listing. Only the curves shown in the 2D Curves window will be transformed to the frequency domain.
4. Click Tools: Frequency Domain: Amplitude from the menu bar. The spectrum of the selected curves are shown in the 2D Curves window. You can also select Tools: Frequency Domain: Phase Shift, Tools: Frequency Domain: Real Part and Tools: Frequency Domain: Imaginary Part.

#### **To view currents in the frequency domain:**

1. Activate the 2D Curves window by clicking anywhere in the window.
2. Check Tools: Time Domain: Current from the menu bar.
3. Select those curves which will be transformed into frequency domain using the Present Curves panel listing. Only the curves shown in the 2D Curves window will be transformed to the frequency domain.
4. Click Tools: Frequency Domain: Amplitude from the menu bar. The spectrum of the selected curves are shown in the 2D Curves window. You can also select Tools: Frequency Domain: Phase Shift, Tools: Frequency Domain: Real Part and Tools: Frequency Domain: Imaginary Part.

#### **NOTE:**

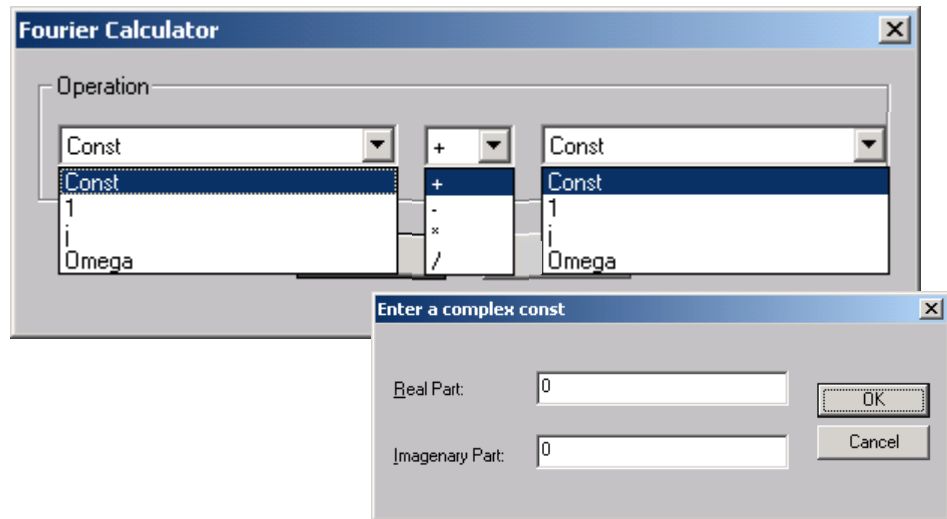
The default parameters for Fourier transforms can be changed through the nFreq and Fmax variables of the Fourier Transformation panel of the Preference dialog box. Refer to [“Tools Menu” on page 3-17](#) for details of the Preferences dialog box.

**Obtaining Frequency Domain Parameters**

Four arithmetic operations (+, -, ×, /) are enabled between any two frequency domain data sets. Using Constant, 1, j, Omega and the combination of them, you can obtain a variety of frequency domain parameters. Select SPDSIM:2D Curves:View:FD Calc... to display a Fourier Calculator popup dialog box. The left and right drop down menu enables the user to select:

- Const — a user input complex number
- 1 — the constant 1
- j — equivalent to the square root of negative 1 ( $\sqrt{-1}$ )
- Omega — the angular frequency

Four operators (+, -, ×, /) are provided to enable addition, subtraction, multiplication or division between the values.



Using Constant, 1, j, Omega and the combination of them, you can obtain a variety of frequency domain parameters.

**Using the Fourier Calculator (an example)**

We will calculate effective inductance as an example of using the Fourier Calculator. Recall, the impedance curve is equal to the V1 curve data divided by the C1 curve data and “effective” inductance (L) equals the imaginary portion of the impedance curve divided by omega ( $\omega$ ). Thus, we will obtain the imped-

ance curve first and then use that curve to obtain the effective inductance curve.

$$\text{impedance curve} \equiv Z = V1 / C1$$

$$\text{effective inductance} \equiv L = I_m \left[ \frac{Z}{\omega} \right]$$

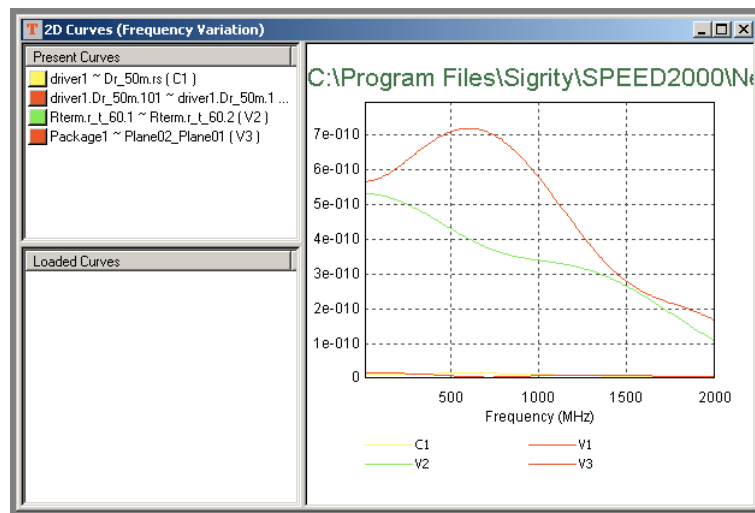
1. Use SPDSIM to simulate a .spd file. Load the file and run SPDSIM to completion of a full simulation cycle.

#### NOTE:

This example will work using the following file delivered on the SPEED2000 cd:

C:\Program Files\Sigrity\SPEED2000\Samples\Example.spd

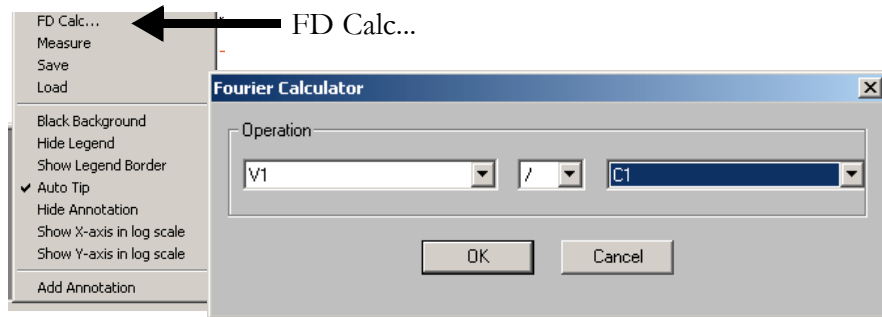
2. Prepare the Fourier Calculator memory. Activate the 2D Curves window and select Tools:Time Domain:Voltage. Right click in the Present Curves panel and select Show All Curves in the popup menu. Observe all of the voltage curves are displayed. Then, select Tools:Time Domain:Current. Right click in the Present Curves panel and select Show All Curves in the popup menu. Observe all of the current curves are displayed.
3. Activate the 2D Curves window and select Tools:Frequency Domain:Amplitude. Observe the 2D Animation panel displays the simulated curve data for both voltage and current curves.



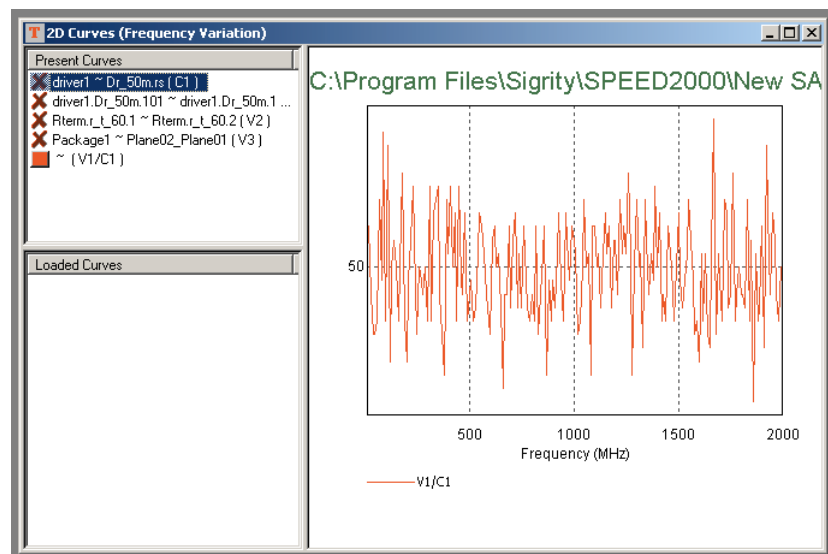
**NOTE:**

If the current and voltage curves are not both displayed, repeat the previous step (which loads the calculator's memory). The memory of the Fourier Calculator must contain the curve data for both voltage and current curves.

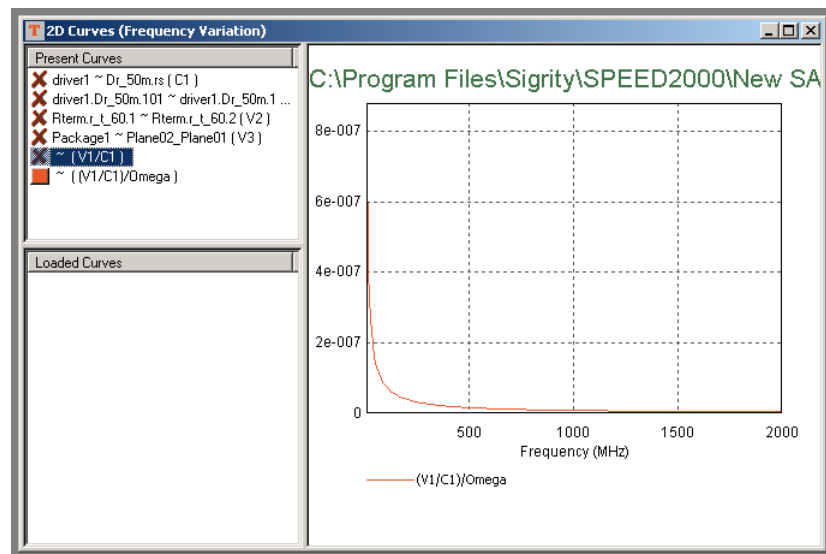
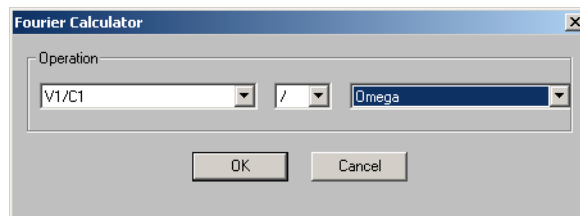
4. Invoke the Fourier Calculator. Right click in any pane of the 2D Curves window (Frequency Variation:Amplitude mode). A popup menu will be displayed. Select FD Calc... The Fourier Calculator is displayed.



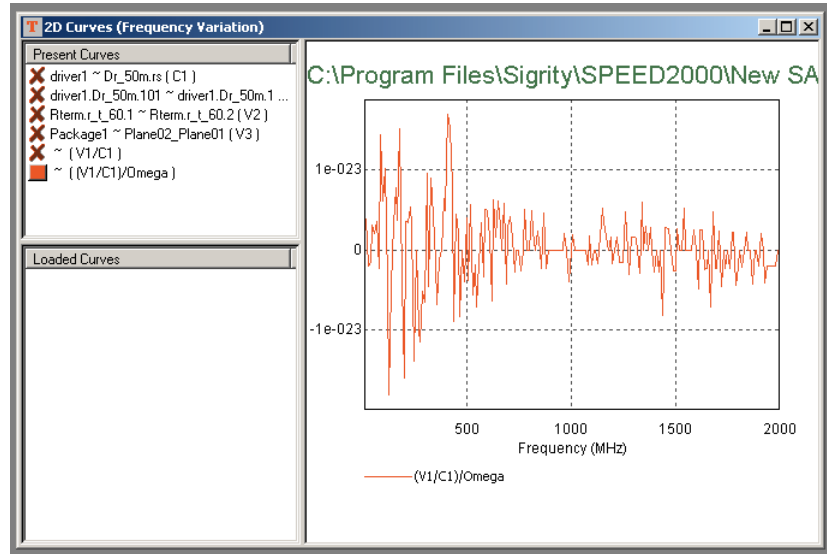
5. Calculate the impedance curve data relative to V1 and C1 (where impedance is the ratio of voltage divided by current). Select V1 as the term in the left side of the calculator. Select C1 as the term in the right side of the calculator. Select the arithmetic operator for divide. Click OK. The Fourier Calculator dialog box is closed. The resulting impedance curve name  $\sim (V1/C1)$  appears in the Present Curves panel. Hide all of the other curves and the impedance curve is displayed in the 2D Animation panel.



6. Now, in accord with the effective inductance formula (shown earlier), divide the impedance by omega( $\omega$ ). Invoke the Fourier Calculator again. Right click in any pane of the 2D Curves window (Frequency Variation:Amplitude mode). A popup menu will be displayed. Select FD Calc... The Fourier Calculator is displayed. Select the new V1/C1 curve as the term in the left side of the calculator. Select “Omega” as the term in the right side of the calculator. Select the arithmetic operator for divide. Click OK . The Fourier Calculator dialog box is closed and the resulting curve name  $\sim ((V1/C1)/\text{Omega})$  appears in the Present Curves panel.



7. Now view the imaginary part of the curve — which is the effective inductance curve. Select Tools:Frequency Domain:Imaginary Part. The imaginary part of the curve  $\sim ((V1/C1)/\Omega)$  is displayed.



***Saving Curve Data into Data Files***

Right Clicking in the 2D Curves window and selecting save from the popup menu will save the curve data of the curves displayed in the 2D Curves window into a text file. Refer to [“SPDSIM Features Overview” on page 3-1](#) for comprehensive details of the menus, toolbars, buttons, dialog boxes and features of SPD-SIM.

**To change the name of a curve:**

1. Double click the curve shown in the 2D Curves window.
2. In the Curve Name field of the Property dialog box, type in the new name for the curve.

***Loading and Plotting Previously Saved Curves***

In the 2D Curves window, the Loaded Curves panel is for loading saved curves from data files. Loading the saved curves enables comparison with the present simulation results. Loaded curve data can also be reorganized, printed and/or saved as bitmap files.

**To load saved curves from data files:**

1. Right Click in the Loaded Curves pane and select load. The Open dialog box appears.
2. Select the data file to be loaded, and click Open. The Load Curve dialog box appears.
3. Select the curves in that data file to be loaded by clicking on the curve names, then click OK.

***Printing the Results***

All window contents such as 2D curves, 3D spatial voltage distributions, and package structures can be printed as shown on the screen. To print, simply click File:Print on the menu bar.



## Calculating Radiation

The radiation calculation is a post-processing feature that enables users to simulate the far field radiation based on the voltages and currents in traces and planes obtained in SPDSIM. You will be able to calculate the electric fields at the location you specified in a similar fashion to the measurement setup and compare the calculation results with FCC requirements. Since the edge radiation of each metal layer is recorded, you will be able to do comparative studies and what-if analyses on package radiation with different plane shapes/sizes, peripheral via shielding strategies, decoupling capacitor placement options and so on.

Radiation calculation in SPEED2000 is a simple two-step operation. You first need to identify in SPDGEN the traces/edges that are to be considered for radiation calculation. Then in SPDSIM, after obtaining voltages and currents through the transient simulation, you need to specify the measurement parameters such as the antenna location, signal period and observation range.

## Selecting Traces and Edges for Radiation Calculation

To calculate the radiation from a trace, it needs to be identified in SPDGEN. Once it is selected, the voltages and currents in this trace will be recorded during the transient simulation in SPDSIM. Only microstrip lines on the top surface (including buried microstrip lines) can be selected for radiation calculation. If a stripline or a microstrip line on the bottom surface is selected, a warning message will occur.



Radiation Select

### ► To select a trace for radiation calculation:

1. In the Layer Selection dialog box, activate the signal layer to be worked on.
2. Click on the Radiation Select button on the tool bar (or click on Edit : Trace : Radiation Select on the menu) to activate the radiation selection process.
3. Click on the Select Single Trace button on the tool bar, then select the traces for radiation calculation. Those traces that will be considered for radiation are now highlighted in bright yellow color.
4. If you want to deselect a trace that is already selected, click on it.

5. When the selection is done, click on the Radiation Select button on the tool bar again to end the process.

In summary, selecting traces for radiation calculation is a very similar process as just selecting a trace. Just make sure that the Radiation Select button is active when you select/deselect traces.



Edge Radiation

► **To select edges for radiation calculation:**

1. Click on the Edge Radiation button on the tool bar (or click on Setup : Edge Radiation on the menu) to enable the edge radiation calculation.

Once the edge radiation option is checked in SPDGEN, the voltages along the edges of each pair of metal planes will be recorded in the simulation by SPD-SIM, and you will be able to view the radiation due to one layer or layers.



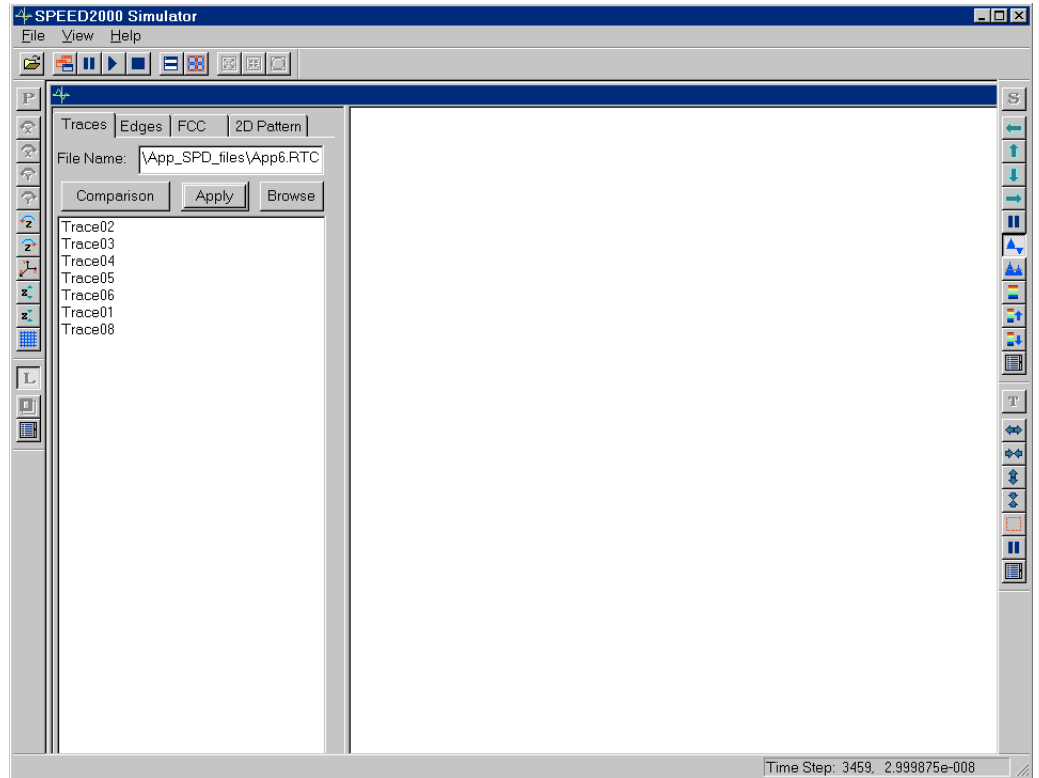
View Radiation

► **To view radiation selections:**

1. Every time when you load a SPD file into SPDGEN, those traces selected for radiation calculation are highlighted in bright yellow color. To disable the highlight, click on the View Radiation button on the tool bar or click on View: View Radiation Trace on the menu.

## Launch Radiation Calculation in SPDSIM

When a transient simulation is finished or stopped, you can launch the radiation calculation by click View: Radiation on the menu. The Radiation window appears as shown below.



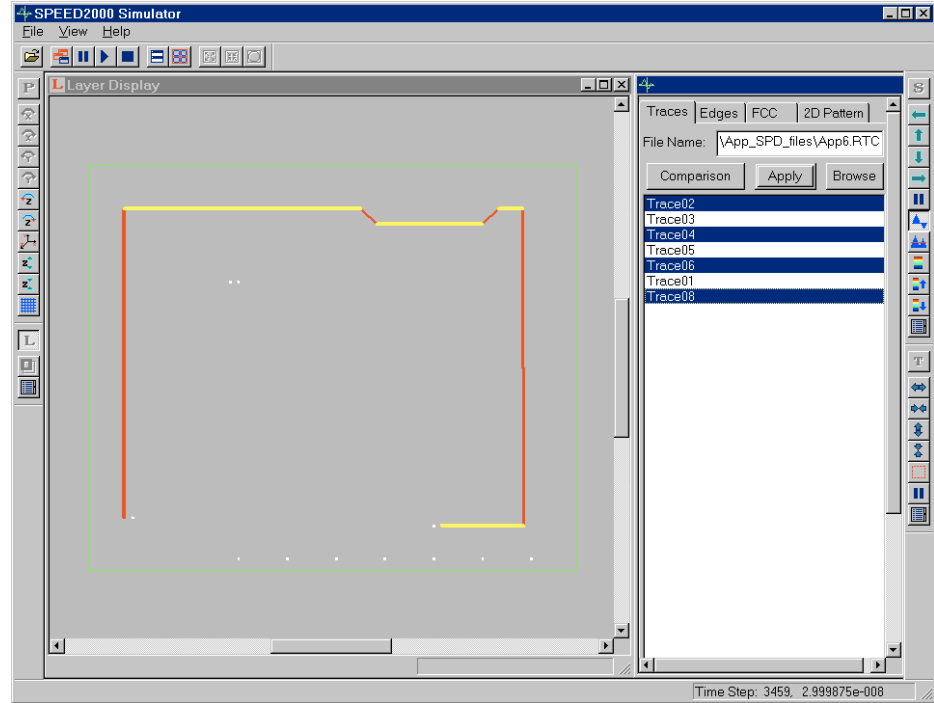
There are four tabs in the left window area for calculation control.

### ***Trace Tab***

In the Traces tab, select the traces to be calculated by clicking on their names. To select multiple traces, hold down the left mouse button, then scroll over the trace names. Traces that are selected for radiation calculation will become highlighted in bright yellow color in the Layer Display window as shown in the following screen. To deselect a trace, hold down the Ctrl key and click on the trace name.

### Edge Tab

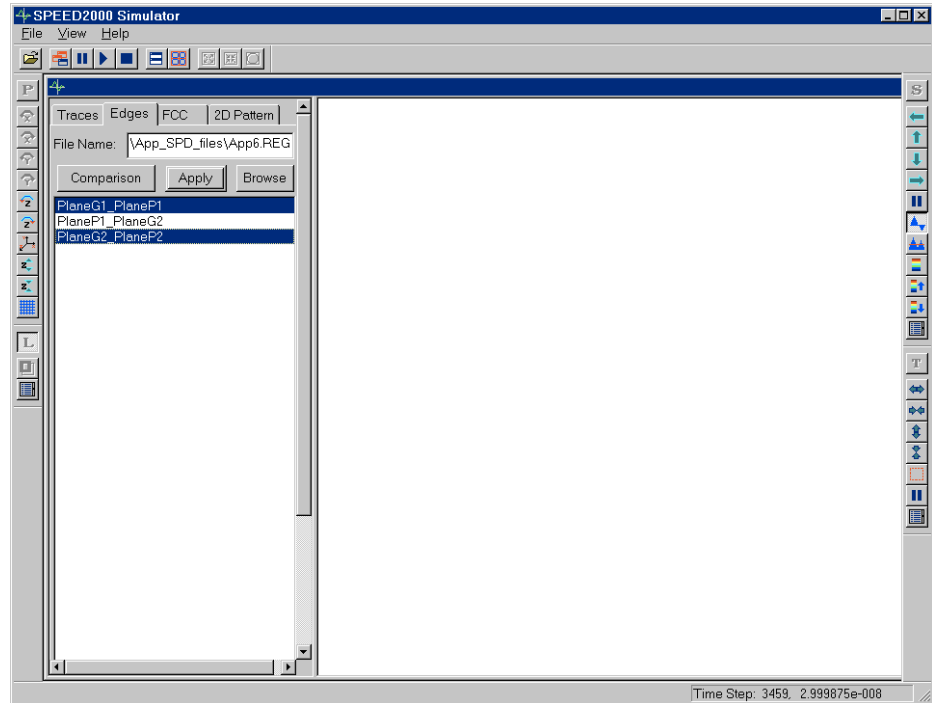
In the Edges tab, select the edges to be calculated by clicking on their names. To select multiple edges, hold down the left mouse button, then scroll over the edge names. To deselect an edge, hold down the Ctrl key and click on the edge name



### ***FCC Tab***

In the FCC tab, two types of calculations can be launched:

1. At a fixed observation point, calculate E vertical field component and E horizontal field component;
2. Find the maximum E field component on a cylindrical surface (with a given distance and height).

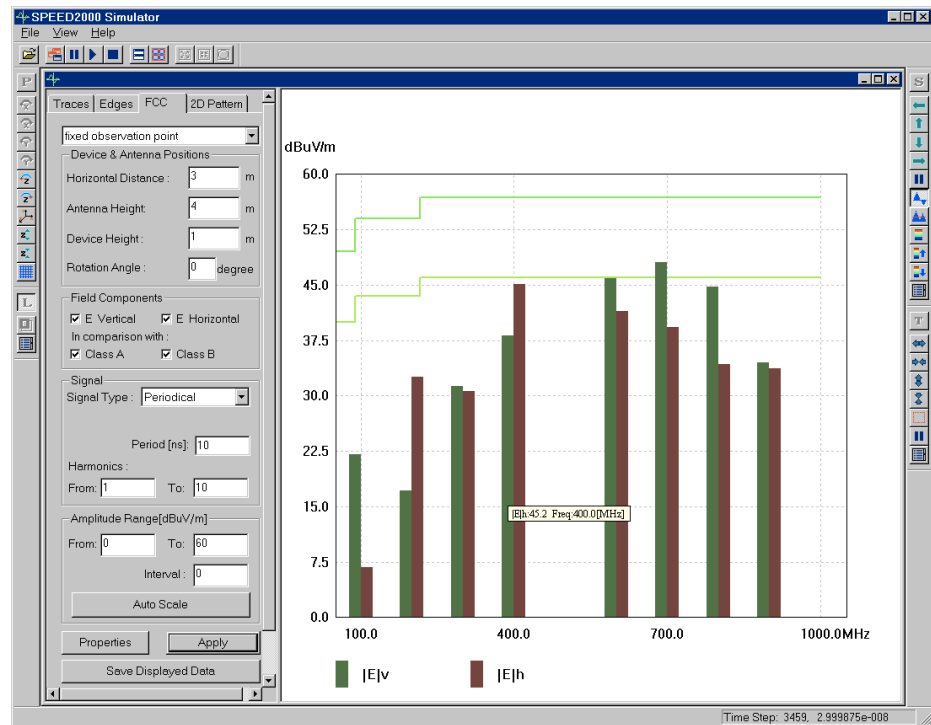


## ► To calculate radiation at a fixed observation point:

1. First select the “fixed observation point” from the pull down list.
2. Enter Horizontal Distance – the distance between Device Under Test (DUT) and the antenna, Antenna Height, Device Height, and Rotation Angle of the package.
3. Check which result need to be displayed, E vertical and/or E horizontal.
4. Check which FCC standard need to be displayed, Class A and/or Class B.
5. Specify the signal type, Periodical or Non-Periodical.

If you select Periodical signal type, then enter signal Period and how many Harmonics need to be shown;

If you select Non-Periodical signal type, then first you need to select the X-axis for frequency to be plotted in either Log Scale or Linear Scale. If you chose Log Scale, you need to specify the plotting parameters such as Number of Frequency Points to be drawn and the Frequency Range; if you chose Linear Scale, you need to specify the frequency Interval for plotting as well as the Frequency Range.



**NOTE:**

If you select periodical signal type, the time domain waveforms obtained from the transient simulation will be transformed to the frequency domain using Fourier series. If you select non-periodical signal type, the time domain waveforms obtained from the transient simulation will be transformed to the frequency domain using Fourier transformation.

6. Enter Amplitude Range for Y-axis, whose unit is always in dBuV/m. After the plot is drawn on the screen, if you want the plot to use automatic scale, click on the Auto Scale button.
7. Click on the Apply button to start the calculation. The results should be plotted in the right window area as shown above.
8. The value of the electric field component at each calculated frequency point can be read out from the plot. Simply move the cursor over to the bar in the plot and the data will be automatically shown on the screen. You may also save the data points into a text file by click on the Save Displayed Data button.

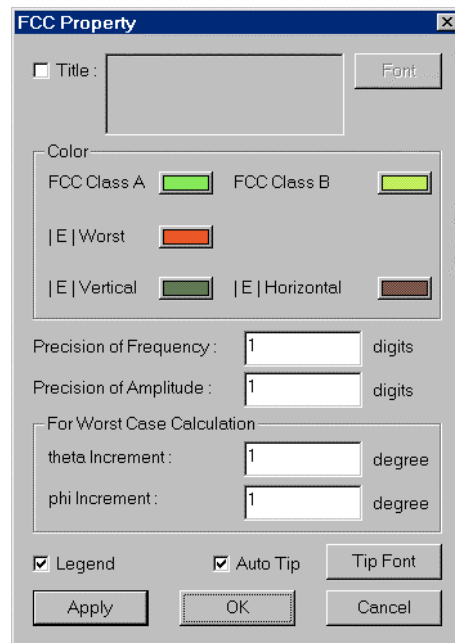
► **To calculate the maximum radiation on a cylindrical surface:**

1. First select “worst case on cylindrical surface” from the pull down list.
2. Enter Horizontal Distance – the distance between Device Under Test (DUT) and the antenna. Enter the Maximum Antenna Height for the searching range. Enter Device Height.
3. Check which FCC standard need to be displayed, Class A and/or Class B.
4. Specify the signal type, Periodical or Non-Periodical.  
If you select Periodical signal type, then enter signal Period and how many Harmonics need to be shown;  
If you select Non-Periodical signal type, then first you need to select the X-axis for frequency to be plotted in either Log Scale or Linear Scale. If you chose Log Scale, you need to specify the plotting parameters such as Number of Frequency Points to be drawn and the Frequency Range; if you chose Linear Scale, you need to specify the frequency Interval for plotting as well as the Frequency Range.
5. Enter Amplitude Range for Y-axis, whose unit is always in dBuV/m. After the plot is drawn on the screen, if you want the plot to use automatic scale, click on the Auto Scale button.
6. Click on the Apply button to start the calculation. The results should be plotted in the right window area as shown below.

7. The value of the maximum electric field component at each calculated frequency point can be read out from the plot, as well as the antenna height and DUT rotation angle under which the maximum value is found. Simply move the cursor over to the bar in the plot and the data will be automatically shown on the screen. You may also save the data points into a text file by click on the Save Displayed Data button.

► **To change options in the FCC plot:**

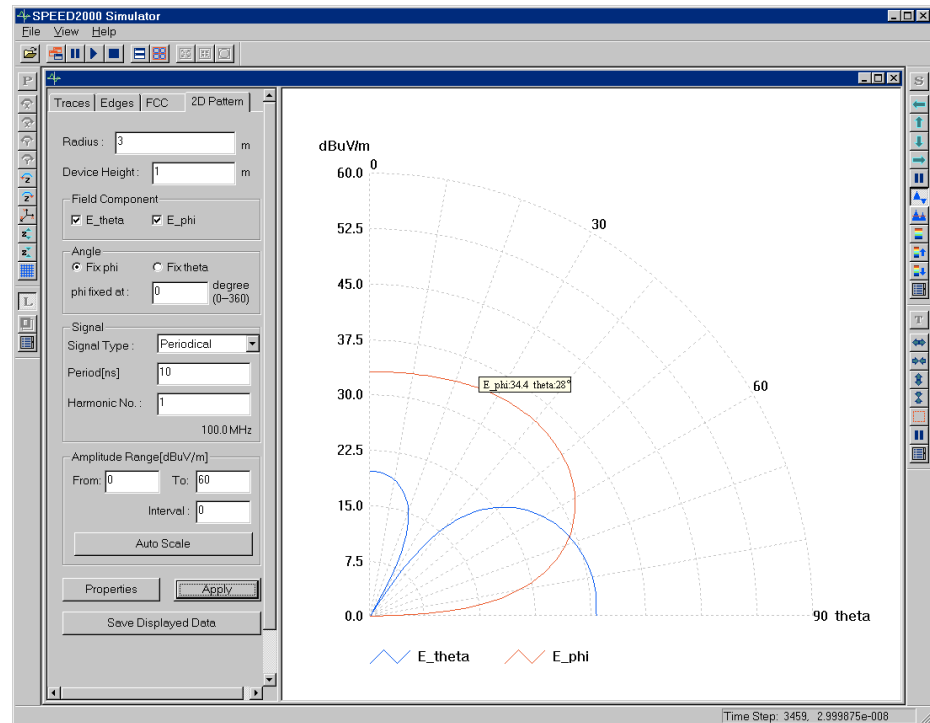
1. To add a title to the plot, change color of the curves or change calculation parameters for the worst case, click on the Properties button. The FCC Property dialog box appears.
2. Change the options in the dialog box, then click OK.





## 2D Pattern Tab

In the 2D pattern tab, you will be able to plot field component  $E_{\theta}$  and  $E_{\phi}$  in polar coordinates.



### ► To plot 2D pattern:

1. Enter Radius.
2. Enter Device Height.
3. Check which Field Component need to be displayed,  $E_{\theta}$  and/or  $E_{\phi}$ .
4. Select Fix phi or Fix theta.

If you select Fix phi, the field component will be plotted for a fixed phi while theta varies from 0 to 90 degrees. Enter the fixed angle for phi.

If you select Fix theta, the field component will be plotted for a fixed theta while phi varies from 0 to 360 degrees. Enter the fixed angle for theta.

5. Specify the signal type, Periodical or Non-Periodical.

If you select Periodical signal type, then enter signal Period and which Harmonic need to be shown;

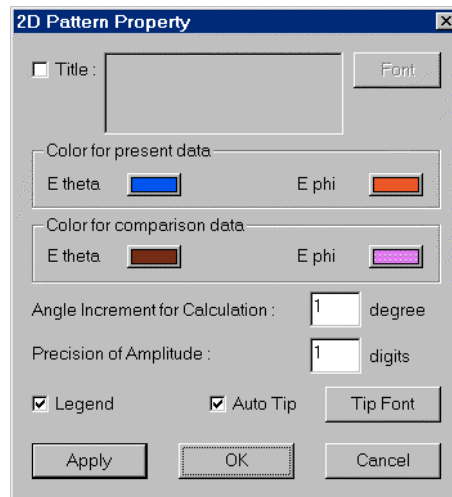
If you select Non-Periodical signal type, you need to specify the Frequency.

6. Enter Amplitude Range for Y-axis, whose unit is always in dBuV/m. After the plot is drawn on the screen, if you want the plot to use automatic scale, click on the Auto Scale button.

7. Click on the Apply button to start the calculation. The results should be plotted in the right window area as shown above.
8. The value of the electric field component at each calculated angle can be read out from the plot. Simply move the cursor over to the curves in the plot and the data will be automatically shown on the screen. You may also save the data points into a text file by click on the Save Displayed Data button.

► **To change options in the 2D Pattern plot:**

1. To add a title to the plot, change color of the curves or change calculation parameters such as Angle Increment, click on the Properties button. The 2D Pattern Property dialog box appears.
2. Change the options in the dialog box, then click OK.



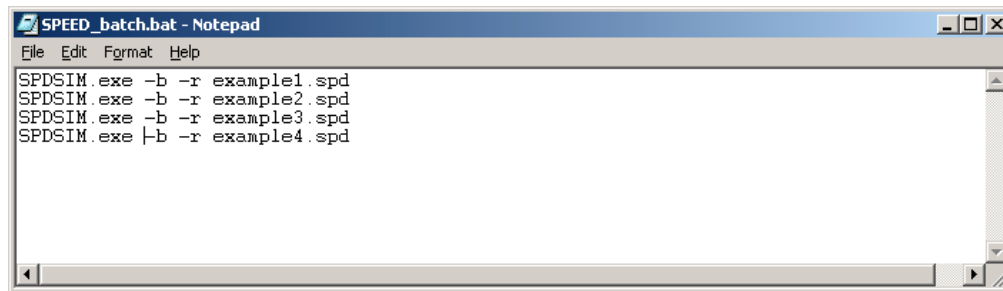
## Printing the Results

All window contents such as FCC plot and 2D Pattern can be printed as shown on the screen. To print, simply click File: Print from the menu.

# Batch Mode Processing

Simulations can be run in batch mode. For example, if you have 20 cases you may want them to run over night. Basically, you create a DOS batch (.bat) file, invoke it and then leave it alone without interaction. Voila! The waveforms at the observation points you *defined in SPDGEN* will be automatically saved into voltage, probability and current data (.cur) files. Refer to [“Establishing Simulation Parameters” on page 6-1](#) for more information on establishing observation points in SPDGEN. Following are two suggestions of ways to configure the .bat file environment to access SPDGEN and SPDSIM and the .spd files of interest:

1. You can copy the SPDSIM.exe and all the .dll files from "C:\Program Files\Sigrity\SPEED2000\Program\SPDSIM.exe" to the local directory where your spd file is located. You then enter: SPDSIM.exe/SPDGEN.exe *switches filename.spd* — where: *filename.spd* is your .spd file of interest; we discuss the “switches” in the following sections [“SPDSIM” on page A-2](#) and [“SPDGEN” on page A-3](#). The example below runs 5 files (example1.spd, example2.spd, example3.spd, example4.spd, example5.spd,)

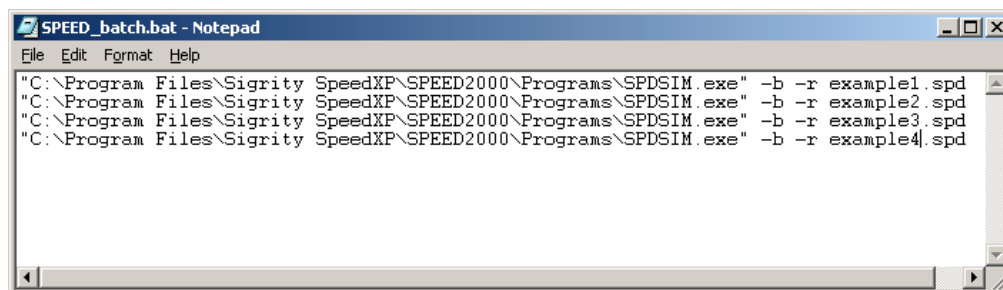


```

SPEED_batch.bat - Notepad
File Edit Format Help
SPDSIM.exe -b -r example1.spd
SPDSIM.exe -b -r example2.spd
SPDSIM.exe -b -r example3.spd
SPDSIM.exe -b -r example4.spd

```

2. The second approach is to give the full path name of the SPDSIM.exe/SPDGEN.exe. (*This way you do not need to copy anything.*) Assuming you are in the local directory where the .spd file of interest resides you can type: "C:\Program Files\Sigrity\SPEED2000\Program\SPDSIM.exe" *switches filename.spd* — where: *filename.spd* is your .spd filename; we discuss the “switches” in the following sections.



```

SPEED_batch.bat - Notepad
File Edit Format Help
"C:\Program Files\Sigrity SpeedXP\SPEED2000\Programs\SPDSIM.exe" -b -r example1.spd
"C:\Program Files\Sigrity SpeedXP\SPEED2000\Programs\SPDSIM.exe" -b -r example2.spd
"C:\Program Files\Sigrity SpeedXP\SPEED2000\Programs\SPDSIM.exe" -b -r example3.spd
"C:\Program Files\Sigrity SpeedXP\SPEED2000\Programs\SPDSIM.exe" -b -r example4.spd

```

## NOTE:

If you encounter problems when attempting to run a batch file (.bat) designed to successively run a series of .spd files on **Windows 98**, some users have had good success by preceding each statement of the batch file with: **start/w**.

## SPDSIM

The batch mode command line switches and parameters for SPDSIM are listed below.

1. Format: **SPDSIM switches filename**
2. Switches: Each switch is prefixed by a '-' (negative) or a '/' (backslash) and they are case insensitive. Unix users use '-' and DOS users '/' for the switch prefix.

- b** switch tells the program to run in batch mode
- s** switch tells the program to run in batch mode with minimized main window.
- n:steps** switch tells the program to output/update the simulation results in the voltage and current data file at each every *steps* steps. If no switch appears the default value is: 100.
- r[:erptm]** switch tells the program to output the selected reports.

e: output execution\_time.log — This detailed log reports the actual time used for each of the main processes of the simulation (reading and processing the .spd file, rearranging partial circuits, etc.) and also the percentage of the total time for each process.

r: output reading\_time.log — This detailed log reports the time used in reading each description line from .spd file. For a large file this can be a very large file.

p: output profile\_spd.log — This detailed log reports the total number of: FDTD layers, nodes, vias, traces, circuits and nonlinear elements.

t: output trace\_extraction.log — This detailed log reports the time used in extracting the parameters for each trace in the model.

m: output memory\_time.log — This detailed log reports: the Computer Name, the Operating System, SPDSIM version/build number, .spd file loaded, Time Steps, Memory Used, Free Memory and CPU Time *for every 1000 time steps of the simulation progress.*

### NOTE:

*-r alone means output all reports.*

3. Filename is the SPD file to be loaded.

### Examples:

- a. **SPDSIM -b -n150 -r app1.spd** — Runs SPDSIM in batch mode; loads app1.spd; after each 150-step, output curves selected in SPDGEN; and output all the reports.
- b. **SPDSIM -b -s -r:ep app2.spd** — Runs SPDSIM in batch mode with minimized main window; load app2.spd; output the execution\_time.log and profile\_spd.log; and after each 100-step (the default), output selected curves.

## SPDGEN

The batch mode command line switches and parameters for SPDGEN are listed below. In batch mode, SPDGEN:

- loads the specified file,
- unionizes the shapes,
- clips the traces,
- identifies special voids,
- segments any traces crossing over split planes,
- performs error checking
- and saves the .spd file.

For more information on the functions performed refer to: [“Working on Shapes” on page 4-29](#), [“To clip traces and unionize shapes:” on page 4-59](#), [“Shape \(and trace\) Process function” on page 4-30](#), [“Working on Shapes” on page 4-29](#), [“Special Voids — Dog-Leg Holes and Small Holes” on page 4-63](#), [“Traces Over “Split Planes”” on page 4-97](#), [“Checking Drawing Errors” on page 4-99](#).

1. Format: **SPDGEN switches filename**
2. Switches: Each switch is prefixed by a '-' (negative) or a '/' (backslash) and they are case insensitive. Unix users use '-' and DOS users '/' for the switch prefix.

-b or -B switch tells the program to run in batch mode.

/b or /B switch tells the program to run in batch mode (Unix).

3. Filename is the .spd file to be loaded.

### Example:

**SPDGEN -b demo.spd** — Runs SPDGEN in batch mode; loads demo.spd.



---

# Index

---

## **Symbols**

(Re)start button 7–5  
.cur  
    Frequency Domain curve file 7–2  
    Probability Distribution curve file 7–2  
    Time Domain curve file 7–2  
.cur file comparison 7–2  
.cur files 7–2  
.spd file 7–2  
.spd File Format Reference 7–3

## **Numerics**

2D Animation area 3–13  
2D curve  
    dynamic display 6–9  
    potential between planes 6–11  
2D Curves  
    Tools menu 3–17  
    View menu 3–15

2D curves  
    viewing 6–13  
2D Curves toolbar 3–23  
2D Curves window 7–4  
    user interface 7–8  
3D spatial voltages  
    dense meshes 6–17

## **A**

About SPDGEN 2–5  
About SPDSIM 3–6  
activate layer drawing/editing 4–7  
add curves for observation 6–12  
add layer 4–8  
Adding new net names 4–20  
adjacent planes  
    potential difference 6–11  
    spatial voltage distribution 6–16  
Adjacent Vias 4–100  
air dielectric layer 4–104

- algorithms
  - Trace Over Split Plane 4–98
- Amplitude 7–12
- analysis
  - “what if” 7–2
  - post processing 7–10
- angular frequency 7–13
- application windows 7–4
- average voltage 6–18
- average voltage amplitude 6–18
- B**
- Background
  - Display in Color 2–19
  - Display Objects 2–19
  - DXF Image 2–19
  - Set Dimension 2–19
  - Set Position 2–19
  - Visible 2–19
- batch mode A–1
- C**
- calculations
  - summary list 7–10
- capacitor 5–1
- change drawing colors 4–13
- Check button (C) 4–101
- Check button (CC) 4–101
- check errors
  - discretization 1–20
  - drawing 1–16, 4–25, 4–99, 6–4
- circle
  - precisely move 4–54
  - precisely size 4–55
- Circuit
  - circuit definition
    - delete 5–7
    - edit 5–8
  - linkage
    - delete 5–14
    - modify 5–14
  - partial circuit
    - delete 5–7
- circuit 5–1
- circuit files
  - external 5–6
- circuit node name 5–3
- circuit nodes TO package nodes 5–9
- circuits
  - override deselected 5–15
- clipped 4–3
- color
  - via display 4–82
- color intensity plot 6–16
- color intensity plot view 7–5
- colors
  - change 4–13
- combine .spd files 4–103
- comparison
  - analytical solutions 6–8
- complex number 7–13
- Component Colors 4–14
- component descriptions 5–3
- computation
  - field domains 4–63
- Computation Parameters
  - establishing 6–2
- computation speed
  - slowdown 6–6
- computation time
  - reducing 6–6
- computational accuracy 6–2
- computational errors 6–2
- computational reference planes 4–63
- conductivity 4–9
- Const
  - complex number 7–13
- constant 1 7–13
- convolution 6–5, 6–6
- copy vias 4–80



- Copy Vias dialog box 4–82
- Couple Line List dialog box 4–96
- coupled lines 2–12, 4–95
  - average separations 4–96
  - crosstalk 4–95
  - electrical properties 4–97
  - identify 4–95
  - internal nodes 4–97
  - parallel length 4–96
  - remove 4–96
  - view 4–96
- cross reference planes 4–98
- crosstalk 2–12, 4–95
- current controlled current source 5–2
- current controlled voltage source 5–2
- current flow
  - direction 6–14
- current source 5–1
- curve files
  - types of 7–2
- D**
- data analysis 7–1
- data flow 7–1
- DC Analysis Option 6–6
- DC loss 6–5, 6–6
- decoupling capacitor 1–XII, 7–12, 8–1
- default (D) 4–93
- default metal thickness 4–9
- default setup 4–7
- delete
  - shape objects 4–46
  - via 4–79
- Delete Illegal Observation Lines 4–3
- delete layer 4–13
- delete nets 4–19
- deselect nets 4–15, 4–16
- deselect object 4–22
- dielectric constant
  - data file 6–6
- dielectric layer 4–11
  - properties 4–11
- dielectric layer (air) 4–104
- Dielectric Loss 6–6
- dielectric loss 4–91
- dielectric lossless 6–6
- dielectric material 4–11
- dielectric model
  - format 4–12
- dielectric models 4–12
- dielectric non-dispersive 6–6
- discontinuity
  - point 4–98
- discretization 6–2, 6–4
- discretization error 1–20, 4–100
- discretize
  - physical structure 6–2
- discretized 6–4
- Dispersion 6–6
- dispersion 4–91
- display mode 7–4
- distributed IR drop (DC drop) on planes 6–6
- Distribution dialog box 6–18
- Dog-Leg Holes 4–63
- draw a point every n time steps 6–12
- Drawing Boxes 4–43
- Drawing Circles 4–44
- drawing colors 2–21
- drawing grid 2–21
- drawing tools 4–24
- DXF files 4–109
- E**
- easily spot errors/warnings 4–99
- Edge Boundary Condition 6–7
  - setting 6–8
- edge radiation 6–19
- edges of metal planes 6–7
- Edit
  - Couple line 2–12

- Delete 2–13
- Find 2–13
- Node 2–12
- Property 2–12
- Shape 2–12
- Stack Up 2–11, 4–38
- Trace 2–12
- Transform Stack Up 2–11
- Transform Stack up By Nodes 2–12
- Via 2–12
- edit
  - several vias 4–78
- electrical properties
  - trace 4–91
- electromagnetic analysis
  - full wave 7–1
- electromagnetic field solver 4–91
- electromagnetic interactions 7–1
- eliminate graphic artifacts 4–30
- Enable Initial DC Analysis Option 6–6
- error
  - minimum distance value 4–102
- Error Checking 4–2
  - mesh on 6–4
- error checking
  - activated 4–102
  - mesh 6–4
- error checking tools 4–99
- error criteria
  - establish 4–102
- error log file 4–102
- error/warning symbols 4–99
- errors 4–100
- errors present 4–102
- establish drawing grid 4–23
- establish error criteria 4–102
- Exiting 4–2
- Export 7–18
- export 7–9
- Extend Vias button 4–83

external circuit files 5–6

## **F**

- Factor 3–22, 7–11
- far field radiation 8–1
- faster animation 6–17
- FCC requirements comparison 8–1
- FD Calc. 3–16, 7–13
- field computation domains 4–63
- File
  - Close 2–9
  - close 4–2
  - Exit 2–3, 2–11, 3–4
  - Merge Circuit Definitions 2–10
  - Merge SPD file 2–10
  - New 2–3
  - Open 2–3, 3–3, 3–9
  - Print 2–10
  - Print Preview 2–11
  - Print Setup 2–3, 2–11
  - Recent File Listing 2–3
  - Recent Listings 2–11
  - Save 2–9
  - Save As 2–9
  - Usage Log File 2–3, 2–11
- file saving options 4–2
- filled pattern 4–6
- Find
  - asterisk 4–113
  - combine search results 4–122
  - Net Manager 4–118
  - patterns 4–116
  - question mark 4–113
  - search criteria 4–113
  - search selected area 4–120
  - searches, constraining 4–113
  - What field 4–114
- find
  - object names 4–114
- Find dialog box 4–113, 4–114, 4–118

- find objects 4–114
- floating node 2–20
- Floating Nodes 4–100
- floating package node 5–15
- Fourier Calculator 3–16, 7–13
- Fourier series 7–8
- Fourier Theory 7–10
- Fourier transform 7–12
- Fourier transform function 3–19
- Fourier Transformation
  - Fmax 3–23
  - nFreq 3–23
- Fourier transforms 7–12
- frequency dependent
  - radiation loss 6–7
- frequency dependent dispersion 6–6
- frequency dependent loss 6–5, 6–6
- Frequency Domain
  - Amplitude 3–19
  - Imaginary Part 3–21
  - Phase 3–20
  - Real Part 3–20
- Frequency/Fourier series display 7–8
- frequency-dependent 4–11

## **G**

- gap 4–97
- gray dot 6–15
- gray dot indicator 4–87
- Gray Unselected 4–16
- grayed vias 4–87
- green vertical lines 4–63
- grid 4–23
- Grid points 6–3
- ground/power bounces 6–18
- group name entity 4–17

## **H**

- Help 2–2, 3–2
- Hide Unselected 4–16

- highlight objects 4–20
- highlight special voids 4–61
- Holes 4–30
- holes
  - dog-leg 4–63
  - small holes 4–63
- hollow via 4–68
- horizontal separation 4–97

## **I**

- IBIS 5–2
- ideal power and ground 1–XI
- identify special voids 4–63
- Imaginary Part 7–12
- impedance 7–8
- impedance curve results 6–7
- Import 7–18
- incomplete .spd file 7–3
- inductor 5–1
- Inner Radius 4–71
- insulating material 4–7
- Intercepted Traces 4–100
- Isolated Nodes 4–100

## **J**

- j equivalent 7–13

## **L**

- layer
  - activate 1–12
  - add 4–8
  - delete 4–13
  - dielectric layer 4–7
  - medium properties 4–11
  - order 4–7
  - plane layer 4–7
  - plane properties 4–10
  - setup 1–11
  - signal layer 4–7
  - signal properties 4–9

- Layer Display
  - View menu 3–40, 3–41
- Layer Display toolbar 3–41
- Layer Display window 3–39, 7–4, 7–5, 7–10
- Layer Selection dialog box 4–5, 4–7, 4–35
- line A 4–95
- line B 4–96
- Line Control field 6–9
- linkage
  - checking 5–14
  - circuit nodes to package nodes 5–9
  - deleting 5–14
  - modify 5–14
- Linkage dialog box 5–2
- linkages 5–3
- linking 5–11
- Loaded Curves 3–13
- loading saved curves 7–18
- loss tangent 4–11
- lossy transmission line 6–7
- Lower Layer 6–16
- lower metal plane 6–12
- Lower Plane 6–17
- lower reference plane 4–98
- M**
  - magnetic walls 6–7
  - managing circuits 5–2
  - maximum voltage swing 6–18
  - Medium Properties dialog box 4–11
  - menu bar 1–10
  - merge 4–103
  - merge .spd files 4–103
  - Merge SPD File dialog box 4–104
  - merged packages 4–83, 4–85
  - merged structure
    - avoid duplication 4–104
  - mesh
    - cells 6–2
    - computation speed 6–2
    - density 6–2
    - error checking 6–4
    - memory requirement 6–2
    - On/Off 6–3
    - select 6–2
    - understanding the 6–3
    - view 4–102, 6–3
  - mesh density 6–2
  - Mesh dialog box 6–2, 6–3
  - mesh lines 6–3
  - Mesh Point 6–4
  - mesh point 6–3
  - mesh points
    - nearest 6–4
  - mesh unit 6–4
  - metal geometries 4–29
  - metal layers
    - separating 4–7
  - metal planes 4–7
  - metal signal traces 4–7
  - metal skin effect loss 4–91
  - Method of Moment 4–91
  - move
    - via 4–75
  - multi-conductor transmission lines 4–95
  - mutual capacitor matrix 5–2
  - mutual inductor 5–2
  - mutual inductor matrix 5–2
- N**
  - names
    - .spd file syntax 4–113
  - native file format 7–3
  - Natural Boundary Condition 6–7
  - nBars 3–22, 7–11
  - negative 1, square root of... 7–13
  - negative box shape 4–43
  - negative circle shape 4–44

- net
  - unnamed 4–15
- Net hierarchical groups 4–17
- Net Manager 4–15
- Net Manager dialog box 4–15
- net name 4–15
- Net Name list 4–17
- Net Names 4–15
- nets 4–15
  - simulating 4–16
  - viewing 4–16
- new file 4–7
- new Shape Definition 4–32
- NewEntity 4–17
- node
  - merge 4–27
  - move 4–26
  - properties 4–26, 4–28
  - select 4–25, 4–27
- Node Properties dialog box 4–26, 4–27
- Node properties dialog box 4–26
- Node Too Close 4–100
- Node Tools 4–24
- noise voltage distribution 6–18
- non-adjacent planes
  - derived spatial voltages 6–16
- non-linear behavioral model 6–6
- normal void 4–67
- null shape definition 4–35
- numerical discretization 6–2
- numerical mesh
  - establishing 6–2
- O**
- object names 4–113
- object properties
  - modify 4–103
- observation
  - specifying for 7–9
- observation symbol 6–12
- Observe button 6–14
- Omega angular frequency 7–13
- Open dialog box 4–1, 4–2
- Outer Radius 4–71
- outline view 4–6
- output file 7–2
- overlapping polygons 4–3, 4–30
- P**
- package
  - ground bounces 6–18
  - modify position 4–105
  - modify size 4–105
  - power bounces 6–18
  - rotation 4–105
- Package Display
  - View menu 3–42
- Package Display toolbar 3–43
- Package Display window 7–4
- Package Information dialog box 4–14
- package linkages 5–3
- package node name 5–3
- package noise voltage distribution 6–18
- partial circuit 5–1
  - delete 5–7
- partial circuit definition 5–1
  - add 5–5
  - delete 5–7
  - editing 5–8
- partial circuit definitions
  - loading external 5–7
- partial circuit library 5–6
- partial circuits
  - add 5–5
- Passivity 6–7
- Passivity Option 6–7
- Patch 4–9
- patch
  - set properties 4–9
- Patch dialog box 4–9, 4–10, 4–36,

- 4–39
  - patches 4–7
  - Pause button 7–5
  - peak voltage 6–18
  - peripheral via shielding strategies 8–1
  - Phase Shift 7–12
  - plane
    - voids 4–98
  - plane edge radiation
    - setting 6–8
  - plane layer
    - set properties 4–10
  - Plane Properties dialog box 4–10
  - Plane properties dialog box 4–34
  - plating thickness 4–68
  - plot 3D display 6–16
  - Polarity
    - direction 6–14
  - Polygon
    - Smoothering Threshold 4–62
  - Polygon Smoothering 4–62
  - Polygon Smoothering Algorithm 4–31
  - Polygon Smoothering Threshold dialog box 4–62
  - Polygon Smoothering Threshold value 4–62
  - Polygon Smoothering/Simplifying function 4–31
  - polygons
    - drawing 4–45
    - simplify 4–62
    - smoothering 4–62
  - positive box shape 4–43
  - positive circle shape 4–44
  - post processing 7–1
  - post processing interface 7–1
  - potential difference
    - adjacent planes 6–11
    - calculate 6–9
    - display 6–9
  - power integrity 1–XI
  - power/ground plane noise 1–XII, 7–10
  - prerequisites 1–XII
  - Present Curves 3–13
  - Present Curves panel 7–8
  - printing results 7–18
  - Probability 3–21
  - probability distribution 7–3
  - Probability Distribution curve 6–18
  - Probability Distribution plot 3–21
    - nBars/Factor 7–11
  - properties dielectric layer 4–11
  - properties ICON 4–103
  - Properties of Extended Via(s) dialog box 4–84
  - properties plane layer 4–10
  - Property icon 4–79
  - Prototype field 5–3
- R**
- radiation
    - 2D pattern tab 8–9
    - antenna location 6–19
    - comparative studies 8–1
    - edge selection 8–2
    - Edges tab 8–4
    - FCC tab 8–5
    - launch calculation 8–3
    - microstrip lines 8–1
    - observation range 6–19
    - select edge 8–2
    - select trace 8–1
    - signal period 6–19
    - trace selection 8–1
    - Traces tab 8–3
    - viewing 8–2
    - what-if analyses 8–1
  - Radiation calculation 6–19
  - radiation calculation 8–1
  - radiation loss 6–7

- radius definition
  - change 4-71
- Radius Name definitions 4-68
- Radius Names 4-68
- Real Part 7-12
- red dot 4-25
- red mesh 4-63
- red-eye icon 4-7
- reference plane
  - rules 4-99
- reference planes 4-63, 4-97
- relative permittivity 4-11
- reload .cur files 7-2
- resistor 5-1
- Resonance 1-XI, 1-XII
- resonant peaks 6-7
- results 7-1
  - comparison between 7-18
  - transient simulation 7-8
- Right-click Pop-Up Menus 3-23, 3-37
- S**
- sampling points
  - coarser 6-17
- saving curve data 7-18
- scaling
  - package 4-105
- segments, trace 4-98
- select nets 4-15
- select node 4-25
- select objects 4-21
- select specific objects 4-21
- Selecting Layer Pairs 3-37
- set display unit 4-22
- Set Grid, Unit and Snap dialog box 4-22
- Setup
  - Distribution 2-17
  - Edge Boundary Condition 2-16
  - Edge Radiation 2-18
  - Ext Ckt Solver 2-18
  - Linkage 2-18
  - Mesh 2-16
  - Simulation View 2-17
    - Delete Illegal Observation Lines 2-17
  - Transient 2-16
- shape
  - add a definition 4-39
  - delete a definition 4-39
  - select a definition 4-38
- Shape Component dialog box 4-47, 4-48
- shape definition 4-29
  - copy to button 4-40
  - delete 4-39
  - patch 4-38
  - select 4-38
  - use 4-33
- Shape Definition dialog box 4-32, 4-48
- Shape Process button 4-30
- Shape Tools 4-24
- Shape Unionizing 4-3
- shapes
  - moving 4-50
- Shift Copy Vias 4-81
- Show Mesh option 4-102
- showing objects 4-20
- signal integrity 1-XI
- signal layer
  - set properties 4-9
- Signal Properties dialog box 4-9, 4-39
- Signal properties dialog box 4-36
- signal traces 4-7
- simulation
  - default values 6-1
  - deselecting circuits 5-15
  - of selected circuits 5-15
  - pause 7-5

- restart 7–5
- stop 7–5
- time step 7–5
- simulation does not converge 6–7
- simulation parameters 6–1
- simulation speed
  - increase 7–5
- simulation time
  - specifying 6–5
- Simulation view
  - circuit current 6–13
  - circuit voltage 6–9
  - package voltage 6–11
  - voltage on the layer 7–6
- Simulation Viewing 6–8
- single transmission line algorithm 4–95
- skin effect
  - plane 6–5
  - transmission line 6–6
- Skip Warnings 4–2
- small holes 4–63
- snap to 2–21
- snap to node function 4–75
- snapping to 4–23
- solid via 4–68
- sort nets 4–19
- spatial animation
  - average voltage 6–18
  - peak voltage 6–18
- Spatial Display
  - View
    - Color Intensity 3–35
    - GreyScheme 3–35
    - Peak 3–35
    - Reset Vmax 3–35
    - V(t) 3–35
    - Vmax- 3–35
    - Vmax+ 3–35
- Spatial Display toolbar 3–36
- spatial distribution
  - data files 7–3
- Spatial Distribution
  - View Menu 3–34
- Spatial Distribution window 7–4
- spatial voltage
  - coarser 3D display 6–17
- spatial voltage distribution 6–16
- spatial voltages
  - derived 6–16
- SPDGEN
  - Toolbar-3D toolbar 2–29
- SPDSIM 2–1, 3–1
- 2D Curves
  - Tools
    - Frequency Domain 3–19
    - Preference 3–22
    - Probability 3–21
    - Time Domain 3–18
- Tools menu 3–17
- View
  - 2D Curve 3–17
  - 3D Distribution 3–17, 3–41
  - FD Calc. 3–16, 7–13
  - Fit 3–15
  - Navigator 3–15
  - Radiation 3–17
  - Scale 3–15
  - Status Bar 3–17
  - Toolbar - 2D 3–17
  - Toolbar - 3D 3–17
  - Toolbar - Layer 3–16
  - Toolbar - Package 3–16
  - Toolbar -Main 3–16
- View menu 3–15
- 2D Curves toolbar 3–23
- 2D Curves window 3–13, 7–8
- File
  - Close 3–9
  - Exit 3–4, 3–11
  - Export 3–9



- OPEN 3-7
- Open 3-3, 3-9
- Print Setup 3-3, 3-10, 3-11
- Recent File Listing 3-4, 3-11
- Reload 3-9
- Usage Log File 3-3, 3-11
- Help
  - About SPDSIM 3-6
- Layer Display
  - Analysis
    - (Re)Start 3-41
    - Initialize Display 3-41
    - Pause 3-41
    - Preference 3-41
    - Stop 3-41
  - View
    - Coordinate 3-40
    - Fit 3-40
    - Show Mesh 3-40
    - Zoom 3-40
  - View menu 3-40, 3-41
- Layer Display toolbar 3-41
- Layer Display window 3-39, 3-42, 7-5, 7-8, 7-10
- Main toolbar 2-5, 3-6
- Package Display
  - toolbar
    - Display Coordinate 3-43
    - Mesh On/Off 3-43
    - X Clockwise 3-43
    - X Counter Clockwise 3-43
    - Y Clockwise 3-43
    - Y Counter Clockwise 3-43
    - Z Clockwise 3-43
    - Z Counter Clockwise 3-43
    - Z Decrease 3-43
    - Z Increase 3-43
- View
  - Coordinate 3-43
  - Fit 3-43
  - Rotate 3-42
  - Show Mesh 3-43
  - Zoom 3-43
  - zScale 3-43
- View menu 3-42
- Package Display toolbar 3-43
- page setup 3-3, 3-10
- print 3-3, 3-10
- Spatial Display
  - View
    - 2D Curve 3-36, 3-40
    - Color Intensity 3-35
    - Fit 3-35
    - GreyScheme 3-35
    - Layer Control 3-36
    - Pause 3-36
    - Peak 3-35
    - Radiation 3-36, 3-40
    - Reset Vmax 3-35
    - Rotate 3-35
    - Status Bar 3-36, 3-40
    - Toolbar - 2D 3-36, 3-40
    - Toolbar - 3D 3-36, 3-40
    - Toolbar - Layer 3-36, 3-40
    - Toolbar - Package 3-36, 3-40
    - Toolbar -Main 3-36, 3-40
    - V(t) 3-35
    - Vmax- 3-35
    - Vmax+ 3-35
    - Zoom 3-35
- Spatial Display toolbar 3-36

- View
    - 2D Curve 3–5
    - Radiation 3–4
    - Status Bar 3–4
    - Toolbar 3–4
  - View menu 3–34
  - Window
    - Arrange Icons 3–12
    - Close All 3–12
    - Fit All 3–12
    - Tile 3–12
    - To Default 3–12
  - special void 4–63
    - manual 4–65
  - special void size criteria 4–64
  - special voids 4–30, 4–61
  - Special Voids Criteria dialog box 4–63
  - Specify Name dialog box 4–32, 4–41
  - speed
    - faster simulation 6–17
    - take much less time 7–4
  - SPEED Curve files (.cur) 7–2
  - SPEED LOG File 2–3
  - speed up 7–5
  - SPICE 5–1
  - split 4–97
  - Split Planes 4–97
  - square symbol 4–85
  - Stack Up
    - edit 4–7
  - Stack Up dialog box 4–7, 4–8, 4–33
  - Stack-up dialog box 4–7
  - stack-up dialog box 4–7
  - start button 7–5
  - starting 7–3
  - statistical calculations 6–18
  - Statistics Curve
    - Factor 3–23
    - nBars 3–22
  - Stop button 7–5
  - sub-circuit definition 5–2
- T**
- technical support 1–XI
  - thickness dielectric medium 4–11
  - thickness metal plane 4–10
  - Time Domain 3–18
    - Current option 3–18
    - Voltage option 3–18
  - Time Interval
    - output results 6–12
  - Time Variation 7–8
  - timesteps 6–5
  - tool
    - node 4–24, 4–25
    - shape 4–24
    - trace 4–25
    - via 4–25
  - Toolbar - 3D 2–29
  - Toolbar - Checking 2–27
  - Toolbar - Circuit 2–28
  - Toolbar - Main 2–26
  - Toolbar - Object 2–26
  - Toolbar - Select 2–27
  - Toolbar - Shape 2–28
  - Tools
    - Check 2–20
    - Display Floating Package Nodes 2–20
    - Display Linked Package Nodes 2–20
    - Options
      - Auto Save 2–22, 4–3
      - Component Color 2–21
      - DXF Scale and Unit 2–21
      - Error Criteria 2–20
      - Fmax(transmission line) 2–20
      - Grid and Unit 2–21
      - Passivity Check 2–23
      - Polygon Threshold 2–22

- Ruler Font 2–22
  - Special Void Criteria 2–22
  - Preference 3–22
  - Shift Copy Vias 2–23
  - top down view 7–4
  - total simulation time 6–5
  - timesteps 6–5
  - trace
    - above all planes 4–99
    - add 1–27, 4–91, 4–92
    - add node 4–94
    - below all planes 4–99
    - between planes 4–99
    - capacitance 4–91
    - capacitance per mm 4–91
    - change properties 4–92
    - characteristic impedance 4–91
    - computational reference planes 4–63
    - connect 1–19
    - connect via 4–94
    - default color 4–9
    - default width 4–9
    - delete 4–93
    - drawing 1–18
    - effective dielectric constant 4–91
    - ending node 4–91
    - environment analyzed 4–98
    - inductance 4–91
    - inductance per mm 4–91
    - insufficient mesh points 4–99
    - isolated 4–95
    - length 4–91
    - move 4–93
    - move node 4–93
    - propagation speed 4–91
    - segmented 4–98
    - select 8–3
    - signal propagation 4–97
    - starting node 4–91
    - via
      - connect 4–75
  - trace break 4–63
  - Trace Clipping 4–3
  - trace drawing 4–92
  - trace no reference plane 4–99
  - Trace Node Placement Error 4–101
  - trace over split plane 4–63
  - Trace Properties dialog box 4–91, 4–93
  - trace radiation 6–19
  - Trace Tools 4–24
  - traces
    - connecting 4–94
    - coupled 4–95
    - frequency dependent loss 6–6
    - introduction 4–91
    - over split planes 4–98
  - transform entire package 4–105
  - Transform Part of the Stack-up dialog box 4–105, 4–106
  - transform procedure 4–107
  - transformed stackup 4–105
  - Transient dialog box 6–5
  - transient simulation
    - Enable Plane Skin Effect 6–5
    - Enable Transmission Line Effect 6–6
  - transient simulation engine 7–1
  - Transient Simulation Parameters 6–5
  - Transient voltage values 6–18
  - translator 4–1
  - translators 1–XI
  - transmission line 5–1
  - transmission line simulation 6–7
- U**
- undo key 4–19
  - uniform and rectangular mesh 6–2
  - unionize shapes 4–55, 4–59
  - unit 4–22
  - Unnamed net 4–15

Upper Layer 6–16  
upper metal plane 6–12  
Upper Plane 6–17  
upper reference plane 4–98

## V

V(t) button 7–5

vertex

- add 4–52, 4–53
- add precisely 4–52
- delete 4–53
- move 4–52
- move precisely 4–51

vertical interconnections 4–68

vertex reduction 4–31

via

- add 4–70, 4–74, 4–75
  - at trace node 4–75
- assigning properties 4–68
- color 4–77, 4–82, 4–83, 4–85
- conductivity 4–77
- default color 4–82
- delete 4–73, 4–79
- edit 4–76
- extend demonstration 4–87
- extending 4–84
- geometry 4–68
- inner radius 4–68
- inserting using existing vias 4–83
- inserting using existing vias and nodes 4–85
- inspect 4–77
- lower node 4–77
- move 4–24, 4–75
- name 4–77
- net 4–77
- outer radius 4–68
- plating thickness 4–68
  - globally specify 4–72
- radius
  - add 4–70

- delete 4–73

- representation 4–85

- snap 4–76

- symbol 4–75

- thickness 4–68

- trace

- connect 4–75

- upper node 4–77

Via Editing dialog box 4–74, 4–76

Via Extension dialog box 4–83, 4–85, 4–87

Via Placement Error 4–100

via plating thickness 4–72

Via Properties dialog box 4–68, 4–69, 4–77

Via Property dialog box 6–15

via radius

- delete 4–69

- edit 4–69

- new 4–69

- view 4–69

Via Radius Map 4–68

Via Radius Map dialog box 4–68

via radius name definition 4–68

via representation 4–75

Via Tools 4–24

vias

- copying 4–80

- edit several at once 4–78

- inherit properties 4–90

- working on 4–68

Vias Properties dialog box 4–78

View

- 2D Curve 3–5

- 3D view 4–4

- 3D View(Layer View) 2–15

- Auto-tip(Layer View) 2–16

- layer selection 4–5, 4–7

- Layer Selection(Layer View) 2–14

- Net Manager 2–14

- Radiation 3–4
- Show All Couple Lines 4–21
- Show All Couple Lines(Layer View) 2–15
- Show Connected Circuit 2–15, 4–21
- Show Grid 4–21
- Show Grid(Layer View) 2–14
- Show Nodes 4–20
- Show Nodes(Layer View) 2–14
- Show Shape 4–21
- Show Shape(Layer View) 2–14
- Show Special Voids 2–15, 4–21
- Show Via Node Contact 4–21
- Show Via Node Contact(Layer View) 2–16
- Status Bar 2–4
- Status Bar(Layer View) 2–15
- Toolbar 2–4
- Toolbar-3D(Layer View) 2–15
- Toolbar-Checking(Layer View) 2–15
- Toolbar-Circuit(Layer View) 2–15
- Toolbar-Main(Layer View) 2–15
- Toolbar-Object(Layer View) 2–15
- Toolbar-Select(Layer View) 2–15
- Toolbar-Shape(Layer View) 2–15
- View Radiation Trace(Layer View) 2–15
- view
  - circuit currents 7–9
  - package currents 7–9
  - spatial distribution 7–6
  - voltage
    - selected location 7–7
- View 3D dialog box 6–16, 6–17
- View Circuit Voltage dialog box 6–9, 6–10
- View Current Dialog box 6–13
- View Current dialog box 6–14, 6–15 trace 6–15
- View Only Active Layer option 5–11
- View Package Voltage dialog box 6–11
- Viewing
  - Probability Distribution 7–10
- viewing
  - 3D spatial voltage 6–8
  - currents 6–8
  - currents frequency domain 7–12
  - package voltage 6–8
  - probability distribution data 7–11
  - voltages frequency domain 7–12
- viewing mode of layers 4–5
- Viewing Parameters 6–8
- viewing parameters 6–1
- viewing scale
  - controlling 7–9
- viewstep 6–5
- virtual relocation 6–4
- Vmax
  - zero 6–17
- Vmin
  - zero 6–17
- void
  - content criteria 4–63
  - manually normal 4–67
  - size criteria 4–63
- Voids 4–30
- voids
  - manually identify 4–63
  - status 4–63
- voltage amplitude
  - average 6–18
  - peak 6–18
- voltage controlled current source 5–2
- voltage controlled resistor 5–2
- voltage controlled voltage source 5–1

Voltage Probabilty Distribution data-  
file 6-1  
voltage source 5-1  
voltages  
    viewing 6-8

## **W**

warnings 4-100  
width  
    individual trace 4-9  
wildcard 4-113  
wireframe animation 7-5  
wireframe plot 6-16

## **X**

X represents 4-85

## **Y**

yellow dot 6-15  
yellow via 4-83

## **Z**

Zoom 4-3