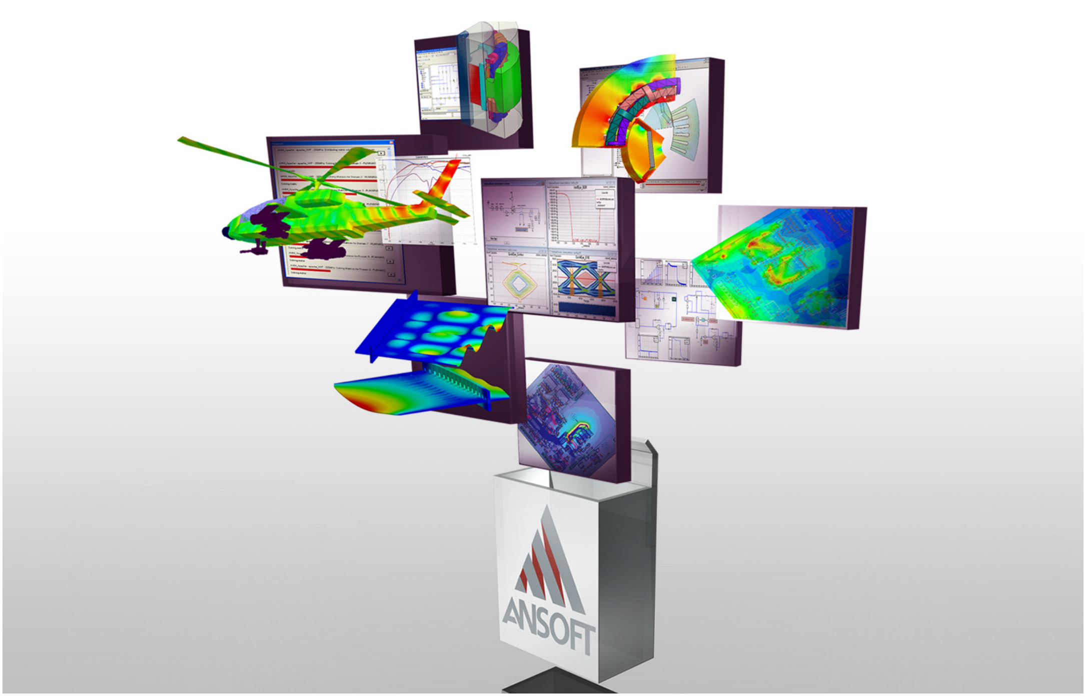


Getting Started with SIwave™: A PCB Model

ANSOFT
ANSYS Product Suite

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
Tel: (+1) 724-746-3304
Fax: (+1) 724-514-9494
General Information: AnsoftInfo@ansys.com
Technical Support: AnsoftTechSupport@ansys.com

November 2010
Inventory: 0000002919

New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Version
1	June 2004	2
2	June 2005	3
3	May 2009	4
4	November 2010	5

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this guide.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands.
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Menu commands are often separated by the ">" symbol. For example, "Click **Draw>Cylinder**".
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press **Shift+F1**" means to press the **Shift** key and the **F1** key at the same time.

Alternate methods or tips are listed in the left margin in blue italic text.

Getting Help

Ansoft Technical Support

To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft website, http://www.ansoft.com/ots/otsLogin.cfm. You will find phone numbers and e-mail addresses for the technical support staff. You may also contact your Ansoft account manager to obtain this information.

All Ansoft software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached. This allows more rapid and effective debugging.

Table of Contents

4. Generating the Solutions

5. Analyzing the Results

1

Introduction

This *Getting Started Guide* is written for SIwave beginners as well as experienced users who would like to quickly familiarize themselves with the capabilities of SIwave. This guide leads you step-by-step through creating, solving, and analyzing the results of simulating nets and planes on a PCB.

By following the steps in this guide, you will learn how to perform the following tasks in SIwave:

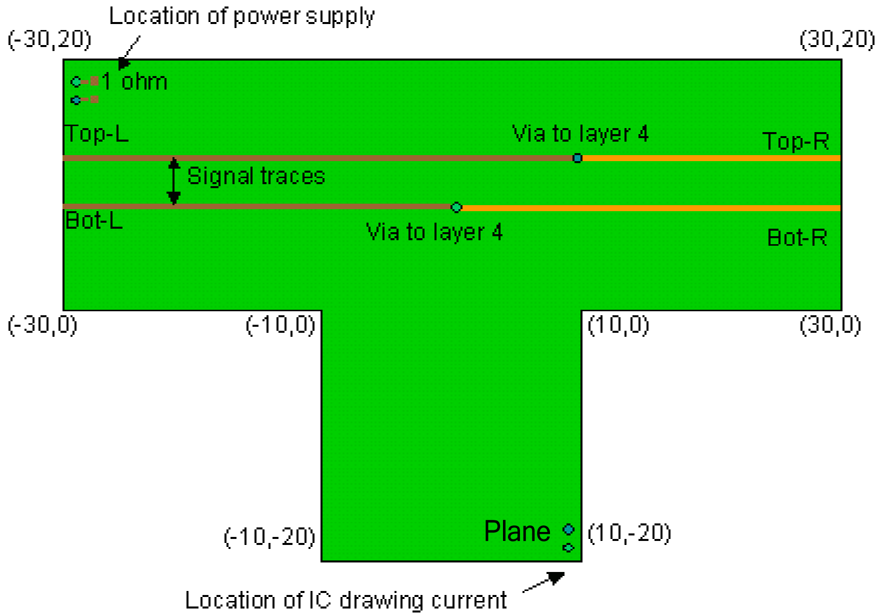
- Draw a geometric model.
- Specify solution settings for a design.
- Validate a design's setup.
- Run an SIwave simulation.
- Create 2D plots of the results.

Estimated time to complete this guide: 60 minutes.

The PCB Model

The PCB model consists of two single-ended transmission lines on an irregularly shaped, 4-layer PCB. Layers 1 and 4 are signal layers, and layers 2 and 3 are planes for power and ground.

The PCB model is shown below.



You will analyze the performance of the nets by injecting signals into the **Top-L** and **Bot-L** terminals. These nets are transmission lines that begin on layer 1 and transition to layer 4. You will examine reflected signals at the same terminals, and transmitted signals at terminals **Top-R** and **Bot-R**.

To analyze the planes, you will calculate the impedance between the planes at the terminal **Plane** in the bottom-right corner of the PCB when a power supply, modeled as a 1-ohm resistor, is connected at the upper-left corner of the PCB.

Expected Results

Ideally, the output signals should represent the input signals. To achieve this, the signal path should be treated like a transmission line whose characteristic impedance is matched to that of the driver and the receiver. Since a via is located on the net, you can expect to see a discontinuity.

For the plane impedance, ideally, you would see the impedance of the power supply at the terminal where the IC is drawing current. If the impedance is too high, an adequate voltage cannot be maintained on the IC at all times.

The implied assumption with the desired behavior is that there is no discontinuity in the signal return path, and, equivalently, that the potential variation between planes is small over the frequency range of interest. Whether this assumption is reasonable depends on the geometry of the PCB.

To determine if the results agree with these expectations, you will examine the results in both the time domain and the frequency domain, and the impedance of the planes versus frequency. You will also examine the resonant modes of the planes to understand the observed behavior.

2 Setting Up the Design

In this chapter you will complete the following tasks:

- Create and save a new project.
- Set up the layer stack.
- Set up the pad stack.
- Set the drawing units for the design.

Estimated time to complete this chapter: 15 minutes.

Open SIwave and Save a New Project

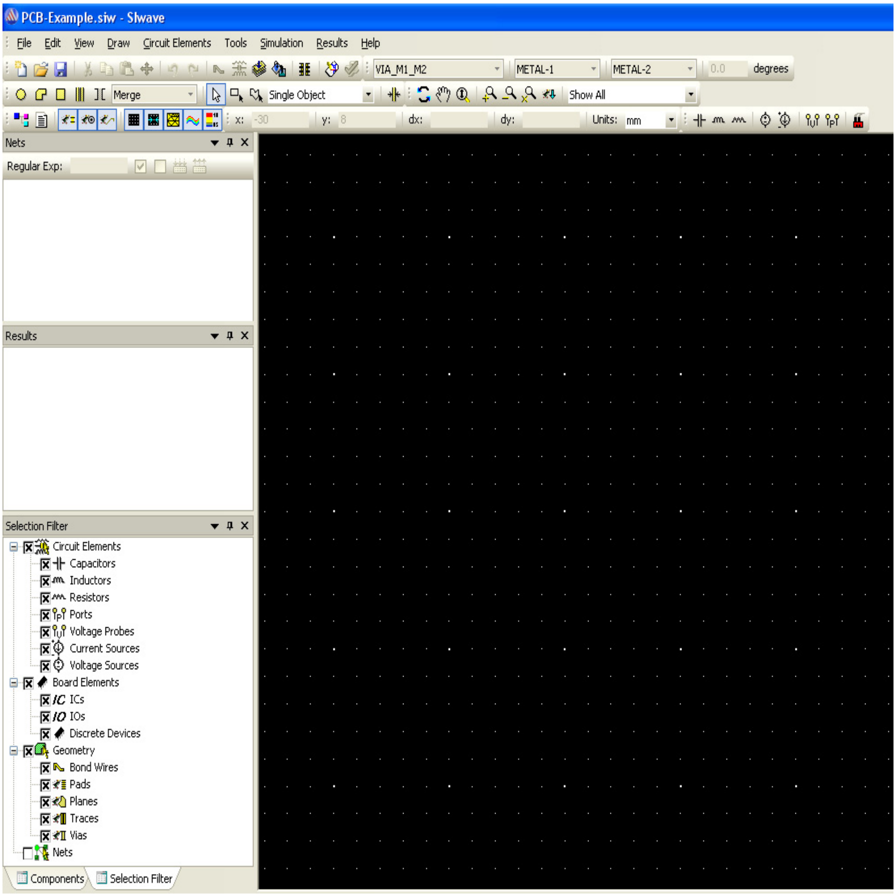
A project file contains information about the geometry, PCB layer stack-up, pad stack, via construction, board materials, and any discrete components.

To start the exercise, launch SIwave and then create a new project file:

1. Double-click the **SIwave** icon on your desktop to launch SIwave.
2. Click **File**>**Save**.

   The **Save As** dialog box appears.
3. Locate and double-click the folder in which you want to save the project, such as C:\Ansoft\SIwave\Project.
4. Type **PCB-Example.siw** in the **File name** box, and then click **Save**.

The project is saved in the folder you selected to the file name *PCB-Example.siw*.



Set Up the Layer Stack

You will need to supply the information about the PCB geometry and materials in the layer stack up.

Set up the layer stack for the PCB:

*In the **Properties** workspace, select **Metal-1**.*

1. Click **Edit>Layer Stack.**

   The **Layer Stack-up Editor** dialog box appears.
2. Change the name of the layer to **Signal-Top**.
3. Leave the default values for the remaining options.
4. Repeat steps through 2 for **Metal-2**, **Metal-3**, and **Metal-4** so that they are called **Ground**, **Power**, and **Signal-Bottom**, respectively.
5. Click **Edit>Layer Stack** to view the new names of the layers in the **Layer Stack-up Editor** window.

Set Up the Pad Stack

In addition to the layer stack, you need to specify information about the pads and vias.

You can set the drill hole for a via by creating or editing a circular pad on the dielectric layer between the two metal layers that the via connects. Therefore, a single via can have different shaft radii depending on the layers that it passes through.

You will now set up the pad stacks for the vias that will be used.

1 Click **Edit>Padstacks.**

   The **Padstack Editor** window appears. Padstack names are listed on the top left, next to the cross-sectional view.

2 Select **Signal-Top** from the Layer list, and verify the following:

   - **Radius** under **Pad Properties** is **0.25mm**.
   - **Shape** under **Antipad Properties** is set to **None.**
   - **Shape** under **Thermal Relief Pad Properties** is set to **None**.

3 Select the other padstacks and layers, and verify that the values correspond to the table below:

Padstack	Layer	Material	Pad Rad. (mm)	Anti-pad Rad.	Thermal Relief Pad Rad. (mm)
VIA_M1_M2	Signal-Top	Copper	0.25	None	None
	Dielectric-1		0.15	None	None
	Ground		0.25	None	None
	Dielectric-2		None	None	None
	Power		None	None	None
	Dielectric-1		None	None	None
	Signal-Bottom		None	None	None

Padstack	Layer	Material	Pad Rad. (mm)	Anti-pad Rad.	Thermal Relief Pad Rad. (mm)
VIA_M1_M3	Signal-Top Dielectric-1 Ground Dielectric-2 Power Dielectric-3 Signal-Bottom	Copper	0.25 0.15 None 0.15 0.25 None None	None None 0.35 None None None None	None None None None None None None
VIA_M1_M4	Signal-Top Dielectric-1 Ground Dielectric-2 Power Dielectric-3 Signal-Bottom	Copper	0.25 0.15 None 0.15 None 0.15 0.25	None None 0.35 None 0.35 None None	None None None None None None None
VIA_M2_M3	Signal-Top Dielectric-1 Ground Dielectric-2 Power Dielectric-3 Signal-Bottom	Copper	None None 0.25 0.15 0.25 None None	None None None None None None None	None None None None None None None

Padstack	Layer	Material	Pad Rad. (mm)	Anti-pad Rad.	Thermal Relief Pad Rad. (mm)
VIA_M2_M4	Signal-Top Dielectric-1 Ground Dielectric-2 Power Dielectric-3 Signal-Bottom	Copper	None None 0.25 0.15 None 0.15 0.25	None None None None 0.35 None None	None None None None None None None
VIA_M3_M4	Signal-Top Dielectric-1 Ground Dielectric-2 Power Dielectric-3 Signal-Bottom	Copper	None None None None 0.25 0.15 0.25	None None None None None None None	None None None None None None None

4 Click **OK** to close the **Padstack Editor** window.

3 Creating the Model

In this chapter you will complete the following tasks:

- Draw the ground plane.
- Copy the ground plane to the power plane.
- Create the signal nets.
- Add ports to the signal nets.
- Add a resistor to model the power supply connection to the plane.
- Add a port to the planes to measure the impedance of the IC connection point.

Estimated time to complete this chapter: 15 minutes.

Create the PCB Layout

The PCB project consists of a four-layer, T-shaped board. Represent the plane on **Ground** by drawing two rectangles. Then, copy the plane to **Power**.

Next, draw signal traces that begin on the top layer and transition to the bottom layer through via holes, and add ports to each terminal.

Finally, add pads on the top layer that connect to the planes through the via holes so that you can simulate the impedance of the planes.

Draw the Ground Plane

You will now draw the T-shaped planes.

1. Click the **Layers** workspace.
2. Click **Ground**.
3. Click **Draw>Rectangle**.
4. Click at (-30, 20) to indicate the starting point.
5. Click at (30, 0) to indicate the end point.
6. Click **Draw>Drawing Mode>Merge**.
7. Draw a second rectangle at (-10, 0) and (10, -20).

Note The two rectangles are automatically merged into a single T-shaped plane.

8. Click **Draw>Rectangle** again to exit the **Drawing Mode**.
9. Click the **Nets** workspace.
10. Select **NET-1** by clicking on the box next to it.
11. Click **Edit>Nets>Change Name**.

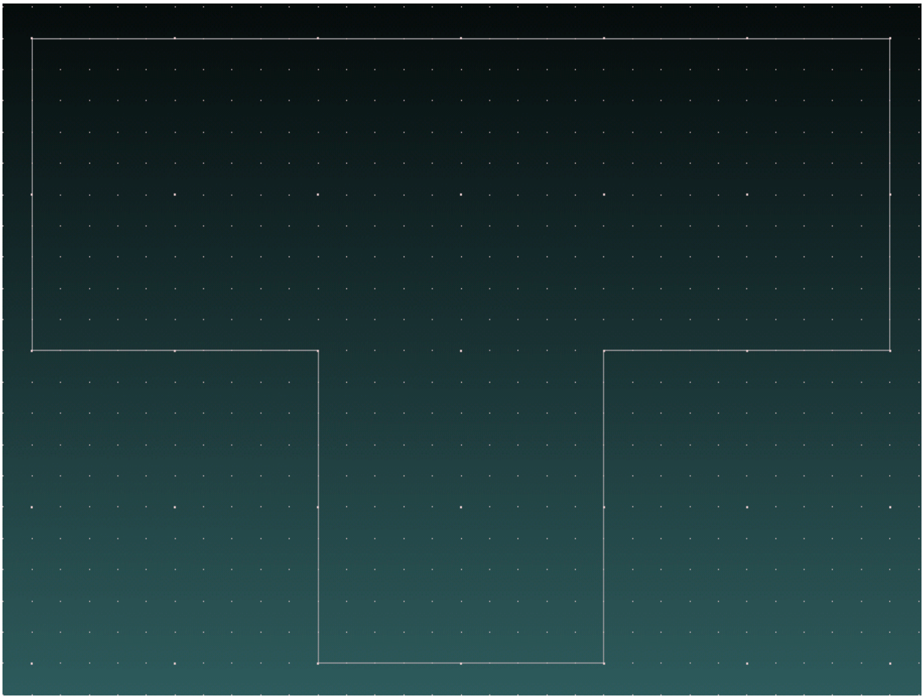
    The **Enter New Net Name** window appears.
12. Enter **Ground** in the **Net Name** box, and click **OK**.
13. Click **File>Save**.

Copy the Ground Plane to the Power Plane

You will now copy the ground plane to the power plane.

1. Click the **Nets** workspace, and select **Ground**.
2. Click **Edit>Copy**.
3. Click the **Layers** workspace.
4. Select **Power** in the list of layers.
5. Click **Edit>Paste**.

6 Move the cursor to position the object so that it overlaps the object on the **Ground** layer.
7 Click the **Nets** workspace.
8 Select **Ground-1** by clicking on the box next to it..
9 Click **Edit>Nets>Change Name**.
   The **Enter New Net Name** window appears.
10 Enter **Power** in the **Net Name** box, and click **OK**.
11 Click **File>Save**.
12 Click the **Layers** tab. Your project should look like the following figure:



Draw First Signal Net

You will now draw the first of the two signal traces.

1 Click **Draw>Set Trace Width**.

The **Set Trace Width** window appears.

2 Type **0.2** in the **Trace Width** box, and click **OK**.

This provides a microstrip line with a characteristic impedance of approximately 50 ohms on the 0.1 mm thick FR-4 layer.

3 Click the **Layers** workspace, and select **Signal-Top**.

4 Click **Draw>Trace**.

5 Click at (-30, 12) to indicate the starting point of the trace on the top layer.

6 Double-click at (10, 12) to indicate the end point.

7 Click **Draw>Via**.

8 Select **VIA_M1_M4** from the pull-down list on the **Via toolbar**.

9 Click at (10, 12).

10 Select **Signal-Bottom**.

11 Click **Draw>Trace**.

12 Click at (10, 12) to set the starting point.

13 Double-click at (30, 12) to indicate the end point.

14 Click the **Nets** tab, and select **NET-1**.

15 Click **Edit>Nets>Change Name**.

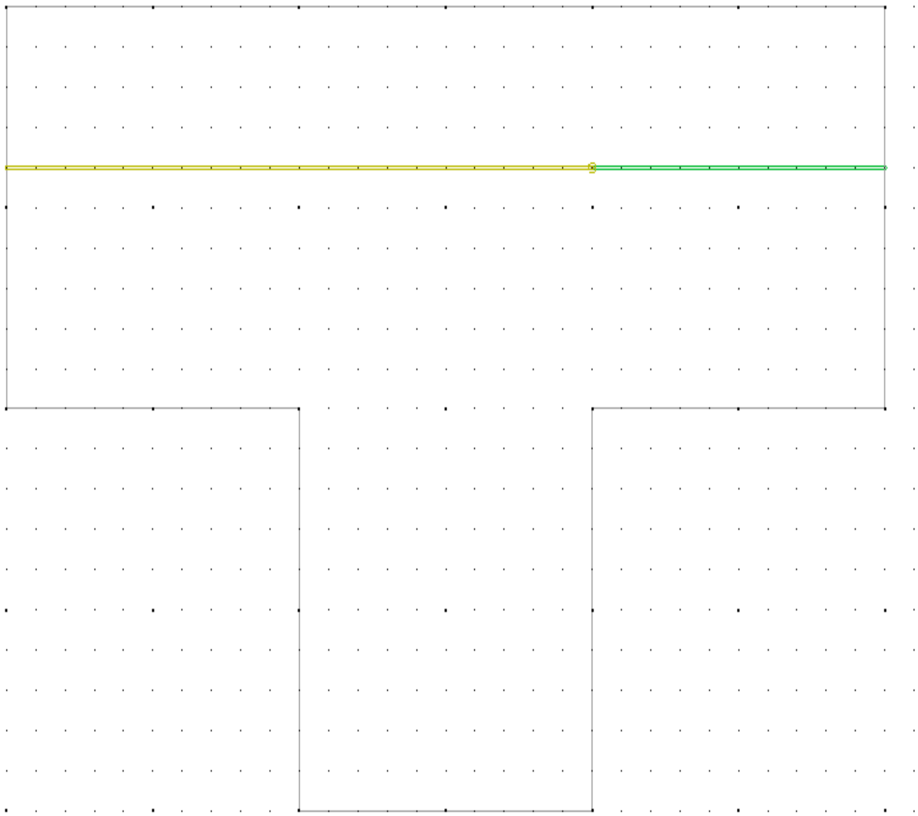
The **Enter New Net Name** window appears.

16 Type **Signal-Top** in the **Net Name** box, and click **OK**.

17 Click **File>Save**.

Your project should look like the following figure:

The background color was modified to white by clicking ***View>Modify Attributes>Gradient Background****.*



Draw the Second Signal Net

You will now draw the second signal traces.

1 Click **Draw>Trace** if you do not have it already selected from the previous step.

1 Click the **Layers** workspace, and select **Signal-Top**.

2 Click at (-30, 8) to indicate the starting point of the trace on the top layer.

3 Double-click at (0, 8) to indicate the end point.

4 Click **Draw>Via**.

5 Select **VIA_M1_M4** from the pull-down list on the **Via toolbar**.

6 Click at (0, 8).

7 Select **Signal-Bottom**.

8 Click **Draw>Trace**.

9 Click at (0, 8) to set the starting point.

10 Double-click at (30, 8) to indicate the end point.

You can also change the net name by right-clicking ***NET-1****, and then choosing* ***Edit Net Name****.*

11 Click the **Nets** workspace.

12 Select **NET-1**.

13 Click **Edit>Nets>Change Name**.

The **Enter New Net Name** window appears.

14 Type **Signal-Bottom** in the **Net Name** box, and click **OK**.

15 Click **File>Save**.

Add Ports to Signal Nets

You will now add ports to the two signal nets so they can be simulated.

1 Click **Circuit Element>Port**.

2 Click at (-30, 12) to define the positive terminal for the port.

3 Click at (-30, 12) to define the reference terminal for the port.

The **Select layers for port terminals** window appears.

4 Select **Signal-Top** for the positive terminal and **Ground** for the negative terminal, and click **OK**.

The **Port Properties** window appears.

5 Type **Top-L** in the **Name** box, and leave the default value for **Reference Impedance**.

6 Click **OK**.

7 Repeat steps 2 through 6 so that you have four ports as described in the table below:

If required, you can edit port values by clicking ***Edit>Circuit Element Parameters****.*

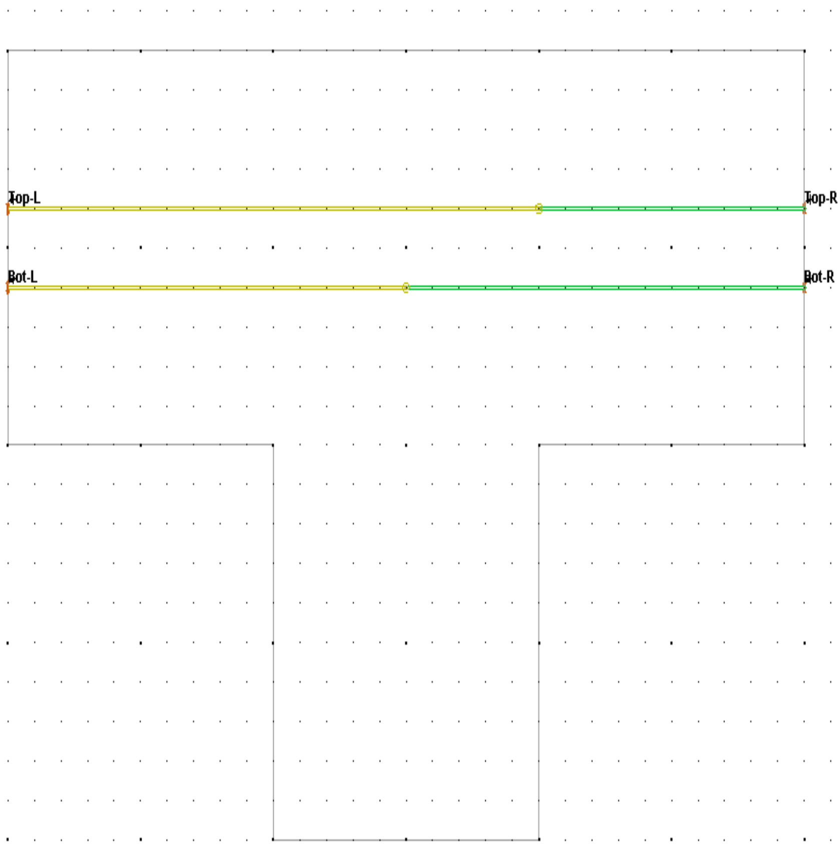
Location				Layer		Port Name	Reference Impedance
Positive Terminal		Negative Terminal					
X	Y	X	Y	Positive	Negative		
-30	12	-30	12	Signal-Top	Ground	Top-L	50
30	12	30	12	Signal-Bottom	Power	Top-R	50
-30	8	-30	8	Signal-Top	Ground	Bot-L	50
30	8	30	8	Signal-Bottom	Power	Bot-R	50

8 Click **Circuit Elements>Port** again to exit the **Drawing Mode**.

9 Click **File>Save**.

10 Click **View>Circuit Elements>Element Names>All on**.

Your project should look like the following figure:



Add the Power-Supply Connection

You will now add a structure to model a power supply connected between the power and ground planes. This structure will consist of two vias connecting to the ground and power planes and extending above the board. Very small planes will be attached to the tops of the vias, and the power supply will be connected to those planes. For the high-frequency simulation, the power supply will be represented as a simple one-ohm resistor.

1 Click **View>Zoom In**.
2 Click on a point slightly above and to the left of the upper-left corner of the board.
3 Click on a second point slightly inside the board. The grid spacing adapts to the new view.
4 Click **Draw>Via**.
5 Select **VIA_M1_M2** from the **Via toolbar**.
6 Click at (-29, 19).
7 Select **VIA_M1_M3** from the **Via toolbar**.
8 Click at (-29, 18).
9 Click the **Layers** workspace, and select **Signal-Top**.
10 Click **Draw>Rectangle**.
11 Click at (-29.2, 19.2) to start a rectangle.

You may want to zoom in, by clicking ***View>Zoom In****.*

12 Click at (-28.0, 18.8) to complete the rectangle.
13 Click at (-29.2, 18.2) to start another rectangle.
14 Click at (-28.0, 17.8) to complete the second rectangle.
15 Click **Circuit Elements>Resistor**.
16 Click at (-29.2, 19.0) and at (-29.2, 18.0) to place the resistor.
The **Select layers for resistor terminals** window appears.
17 Select **Signal-Top** for both the positive and negative terminals, and click **OK**.
The **Set Resistor Parameters** window appears.
18 Type **1** in the **Resistance** box, and leave the default value for the other options.
19 Click **OK**.
20 Click **Circuit Elements>Resistor** to exit this option.
21 Select **View>Fit All**.
22 Click **File>Save**.

Add the IC Connection

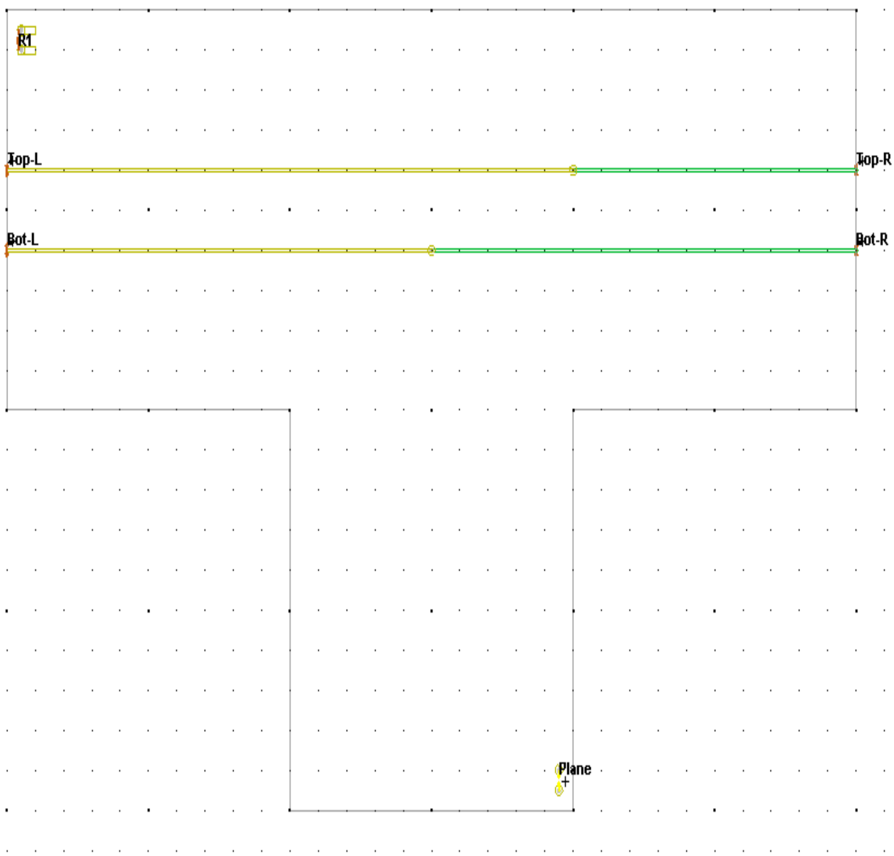
You will now add a port at the IC location to examine the plane impedance “seen” by the IC. This port will connect to the vias, which, in turn connect to the planes.

1. Click **View>Zoom In**.
2. Click on a point slightly below and to the right of the lower-right corner of the board.
3. Click on a second point slightly inside the board. The grid spacing adapts to the new view.
4. Click **Draw>Via**.
5. Select **VIA_M1_M2** from the **Via toolbar**.
6. Click at (9, -19) to create the ground connection.
7. Select **VIA_M1_M3** from the **Via toolbar**.
8. Click at (9, -18) to create the power connection.
9. Click **Circuit Elements>Port**.
10. Click at (9, -19.0) and at (9, -18.0) to place the port, snapping to the vias.

    The **Select layers for port terminals** window appears.
11. Select **Signal-Top** for both the positive and negative terminals, and click **OK**.

    The **Port Properties** window appears.
12. Type **Plane** in the **Name** box, and leave the default value for the reference impedance.
13. Click **OK**.
14. Click **Circuit Elements>Port** to exit this option.
15. Select **View>Fit All**.
16. Click **File>Save**.

Your project should look like the following figure:



4 Generating the Solutions

In this chapter you will complete the following tasks:

- Generate the S-parameters for the signal nets.
- Generate the TDR and TDT responses for the signal nets.
- Display the impedance of the planes on a log-log plot.

Estimated time to complete this chapter: 15 minutes.

Solve for the Performance of the PCB

Now you will analyze the performance of two aspects of the PCB. The first is the signal-integrity characteristics of the signal nets. You will examine these in the frequency domain using s-parameters, and in the time domain by looking at the TDR and TDT responses.

Next, review the power-integrity characteristics of the planes by examining the impedance between the power and ground planes versus frequency.

Analyze Performance of the Signals Traces

You will now analyze the performance of the signal nets.

1. Click **Simulation>Compute S-, Y-, Z-parameters**.

   The **Compute SYZ-parameters** window appears.
2. Specify the following values:

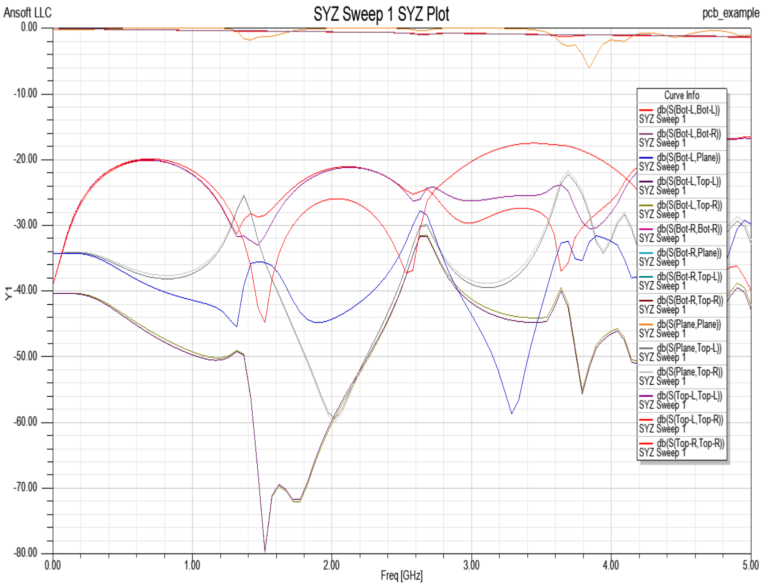
Start Frequency	5e6
Stop Frequency	5e9
Number of Points	1000

3. Click **OK**.

   SIwave simulates the model with various values in the specified range. After the solution is generated, the results are displayed in the **Results** workspace.
4. Double-click the sweep.

   The **SYZ Parameter Plot Generation** dialog appears.
5. Leave the default values for all options, and click **Create Plot**.

SIwave Reporter is launched, and the S-parameter plot is generated.



Generate the TDR Plot

The TDR (time-domain reflectometry) plot shows the impedances encountered by the signal as it travels along the line. These plots help determine the impedance that the signal encounters along its path.

1. In the **SIwave Reporter**, click **SIwave>Results>Create Standard Report>Rectangular Plot**.

   The **Report** window appears.
2. From the **Domain** pull-down list on the left, select **Time**.
3. Click the **TDR Options** button.

   The **TDR Options** window appears.

   Do the following:

   a. Click the **Step** radio button, and then type **35ps** in the **Rise Time** box.

   b. Type **1e-8** in the **Maximum Plot Time** box.

   c. Type **1e-11** in the **Delta Time** box.

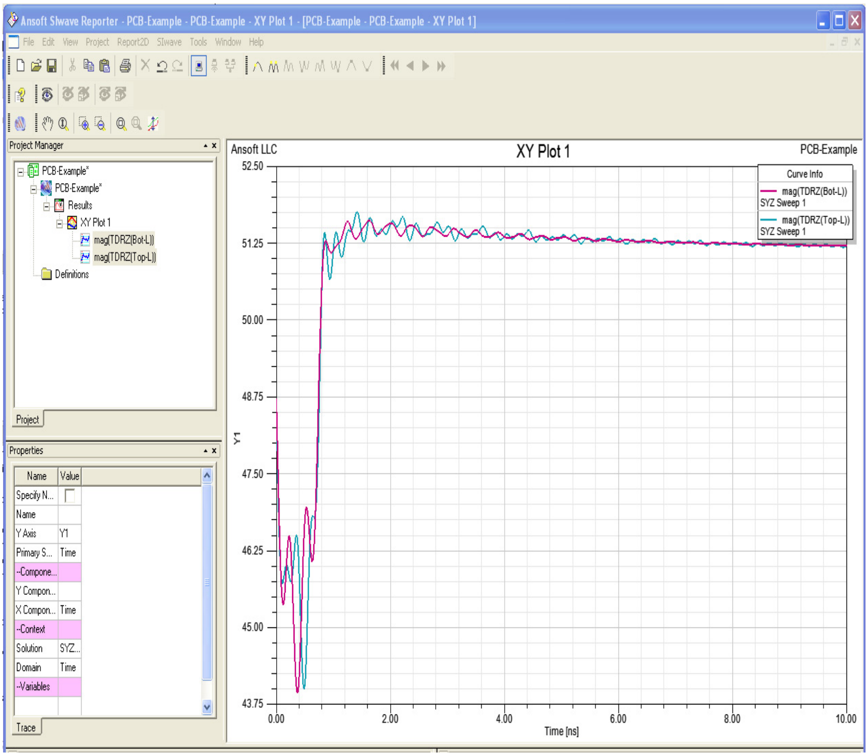
   d. Leave the default values for all other options, and click **OK** to close this window, and return to the **Report** window.
4. In the **Category** list, select **TDR Impedance**.
5. In the **Quantity** list, do the following:

   a. Click **TDRZ(Bot-L)**.

   b. Press **Ctrl**, and click **TDRZ(Top-L)**.
6. In the **Function** list, verify that **mag** is selected.
7. Click **New Report**.

   A trace represents a line connecting data points on the plot.
8. Click **Close**.

A graph showing the TDR response for the traces is generated. The plot is listed as **XY Plot1** under **Results** in the project tree.



The plot shows that the transmission line has an impedance near 46 ohms. Due to the limited frequency sweep, the resolution is not good enough to distinguish the via. The steady-state impedance is approximately 51 ohms, as the power supply and the vias add to the 50 ohm termination.

Generate the TDT Plot

The TDT (time-domain transmission) plot shows how a voltage step with a specified rise time is affected by the impedance discontinuities it has encountered as it traveled the length of the transmission line.

1. Click **SIwave>Results>Create Standard Report>Rectangular Plot**.

   The **Report** window appears.
2. From the **Domain** pull-down list on the left, select **Time**.
3. Click the **TDR Options** button.

   The **TDR Options** window appears. Do the following:

   a. Type **35ps** in the **Rise Time** box.

   b. Type **1e-8** in the **Maximum Plot Time** box.

   c. Type **1e-11** in the **Delta Time** box.

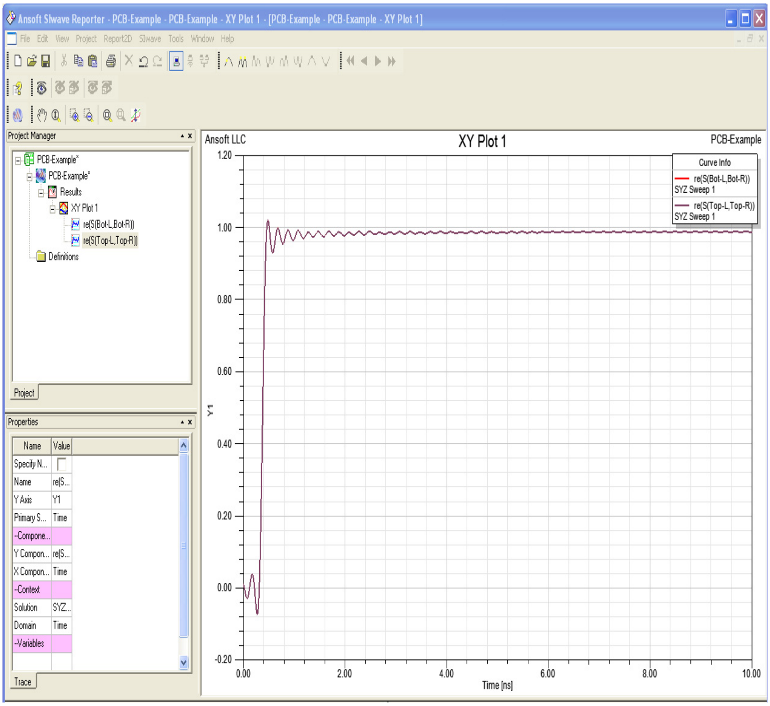
   d. Leave the default values for all other options, and click **OK** to close the window.
4. In the **Category** list, verify that **S Parameter** is selected
5. In the **Quantity** list, do the following:

   a. Click **S(Bot-L,Bot-R)**.

   b. Press **Ctrl**, and click **S(Top-L,Top-R)**.
6. In the **Function** list, verify that **re** is selected.
7. Click **New Report**.

   A trace represents a line connecting data points on the plot.
8. Click **Close**.

A graph showing the TDT response for the traces is generated.



The observed delay time matches the expected delay time for a 60 mm long line on FR4_epoxy.

Analyze Performance of the Planes

You will now analyze the performance of the planes.

1 Click **SIwave>Results>Create Standard Report>Rectangular Plot**.

The **Report** window appears.

2 In the **Category** list, click **Z Parameter**.

3 In the **Quantity** list, click **Z(Plane,Plane)**.

4 In the **Function** list, click **mag**.

5 Click **New Report**.

6 Click **Close**.

The impedance of the planes versus frequency is displayed.

7 Double-click the y-axis.

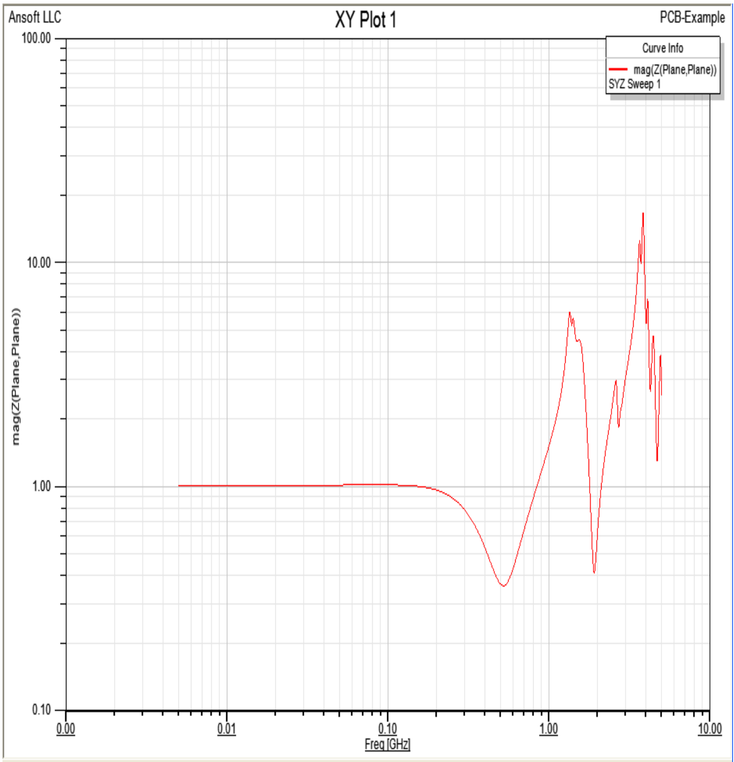
The **Contrast Properties** window appears.

8 Click the **Scaling** tab.

9 Click the **Log** radio button on the right side of the window, and then click **OK**.

10 Repeat steps 8 and 9 for the x-axis.

A log-log impedance plot of the planes is displayed.



At a low frequency, the one-ohm resistance of the power supply is modeled.

11 Click **File>Save** to save this plot.

12 Click **File>Exit** to close the Ansoft SIwave Reporter.

5 Analyzing the Results

In this chapter you will complete the following tasks:

- Calculate the resonant modes of the PCB.
- Compare the results of the resonant mode calculation with the s-parameters calculation.

Estimated time to complete this chapter: 15 minutes.

Analyze the Behavior of the PCB

In this section, you will examine the results obtained from the simulation more closely to see how they match the performance you expected.

Characteristics of the Two Signal Nets

You will compare the performance of the signal nets. Looking at S21 for **Signal-Top** and **Signal-Bottom** shows that their performance significantly differs as frequency increases. This seems surprising since they are located relatively close to each other, with the only difference being the position of the via along the trace. Even so, the nulls in the frequency domain responses are different. There are frequencies where both nets exhibit nulls, although the depths of the nulls are different (i.e. 2.626 hz).

The path of the return current for each net is not obvious. Ideally, the return current should follow a path very close to the signal path. Since the planes are solid, the only location where there might be a discontinuity in the return path is near the via where the return path changes from the ground plane to the power plane. A difference in the potential between the two planes indicates that the current in the return path encounters a discontinuity due to a high impedance.

At low frequencies, the voltage between the planes should be very small since they are connected together with a low-value resistor. However, as the frequency increases, and the dimensions of the plane approach a significant fraction of a wavelength, you can set up resonant modes. SIwave can calculate the relative voltage between the planes for each of these modes.

Calculate Resonant Modes

You will now calculate the resonant modes of the PCB to determine if these are responsible for the difference in performance of the two signal nets.

1. Click **Simulation>Compute Resonant Modes**.

   The **Resonant Mode Solution Options** window appears.
2. Type **10** in the **# of Modes to Compute** box.
3. Leave the default values for all other options, and click **OK**.

After the simulation finishes, the results are listed under the **Results** workspace. The resonant modes are located at nearly the same frequencies as the nulls in the S21 responses for the two nets.

4 Click **Results>Resonant Mode**. Then, select the simulation name and click **View Results**.

The **Resonant Modes** dialog box appears.

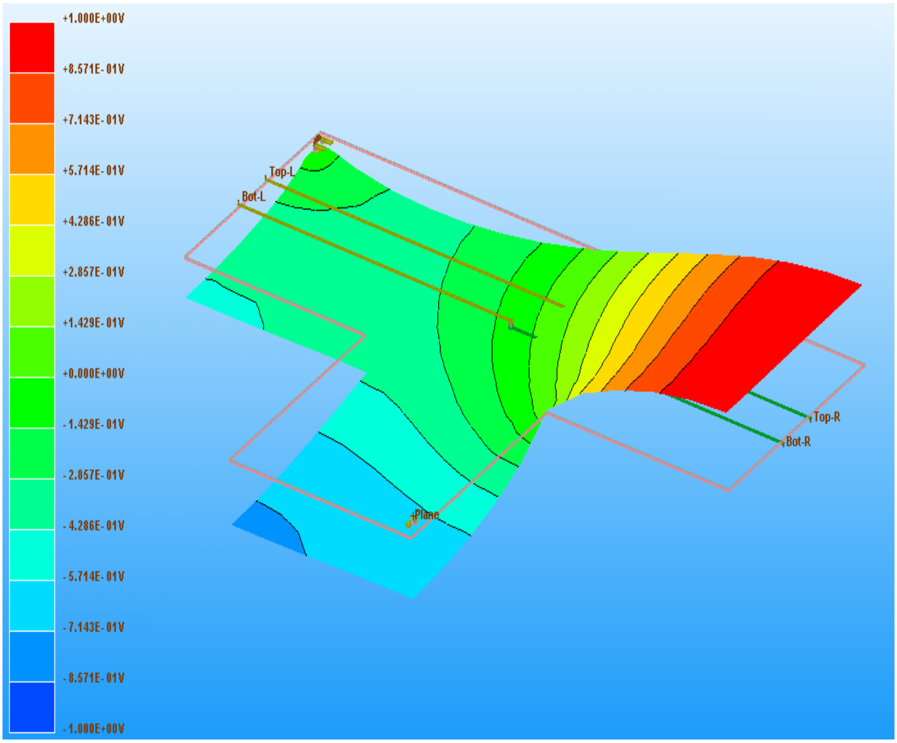
5 In the **Plot voltage difference between planes on** box:

a. Select **Ground** from the first pull-down list.

b. Select **Power** from the second pull-down list.

c. Click **Compute**.

After the simulation finishes, your desktop should look like the following:



6 Click the **Phase Animation** button in the **Resonant Mode Results** window.

The **Phase Animation** window appears.

7 Click the **Generate Frames** button.

The frames are generated and displayed in the window.

8 Click the  icon to continuously loop through the simulations. To stop, click the  button.

9 See the different views of the model in the desktop:

- Click **View>Zoom In**.
- Click **View>Rotate**.
- Click **View>Pan**.

10 Click **Close** in the **Phase Animation** window.

11 Click **Close** in the **Resonant Modes** window.

Compare S21 Responses

You will now analyze the performance of the signal nets, and then generate the S21 plot.

1 Click **Results>SYZ**. Then, select the simulation name and click **Plot Magnitude/Phase**.

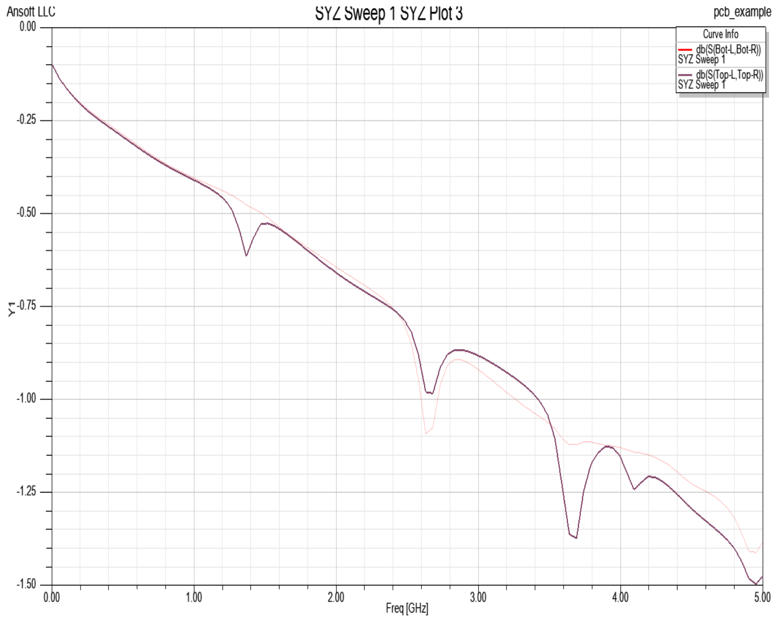
The **SYZ Parameter Plot Generation** dialog box appears.

2 Select **Bot-L, Bot-R.**

3 Select **Top-L,Top-R.**

4 Leave the default values for all other options, and click **Create Plot**.

SIwave Reporter is launched. A graph showing the S21 response for the traces is generated.



5 Click **Close** to close the **SYZ Parameter Plot Generation** dialog box.

Close the Project and Exit SIwave

Congratulations! You have successfully completed *Getting Started with SIwave: A PCB Model*! You may close the project and exit the software.

1. Click **File>Save**.
2. Click **File>Exit**.

Index

S

T

V