

# **Getting Started with Slwave: A PCB Model**



ANSYS, Inc. Southpointe 2600 Ansys Drive Canonsburg, PA 15317 ansysinfo@ansys.com https://www.ansys.com (T) 724-746-3304 (F) 724-514-9494

Release 2022 R2 July 2022

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

#### **Copyright and Trademark Information**

© 2002-2022 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXIm and FLEXnet are trademarks of Flexera Software LLC.

#### **Disclaimer Notice**

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

#### **U.S. Government Rights**

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

#### **Third-Party Software**

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

#### Conventions Used in this Guide

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

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Command font is used for:
  - Command line prompts that should be typed exactly as written.
  - Script examples.
- Bold type is used for the following:
  - Names of windows, workspaces, menu commands, and options.
    - Menu commands are often separated by angle brackets. For example, **File > Open**.
  - Labeled keys on the computer keyboard. For example, Enter.
- Italic type is used for the following:
  - Emphasis.
  - Publication titles.
- 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, while holding it down, press the F1 key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

#### **Getting Help: Ansys Technical Support**

For information about Ansys Technical Support, go to the Ansys corporate Support website, <u>http://www.ansys.com/Support</u>. You can also contact your Ansys account manager in order to obtain this information.

All Ansys 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, including software files as applicable. This allows more rapid and effective debugging.

# **Table of Contents**

Table of Contents	Contents-1
1 - Introduction	1-1
The PCB Model	1-1
Expected Results	1-2
2 - Setting Up the Design	2-1
Importing and Saving the Project	2-1
Setting PCB Element Visibility	
Viewing the Layers Workspace and Layer Stackup	2-5
Viewing the Layer Stackup	2-6
Identifying Power and Ground Nets	2-7
Running a Validation Check	2-8
3 - Resonant Modes Analysis	
Running the Resonant Modes Analysis	3-1
Viewing Resonant Modes Analysis Results	3-2
4 - Slwave SYZ Analysis	4-1
Defining Pin Groups for GND and VCC	4-1
Defining a Port Between Pin Groups	
Generating SYZ Parameters	4-5
Viewing Impedance Response	4-8
5 - PSI SYZ Analysis	5-1
Generating SYZ Parameters using PSI	5-1
Viewing Impedance Response	5-5
6 - PSI AC Current Analysis	6-1
Creating a Voltage Source	6-1
Calculating AC Currents	6-3
Viewing AC Currents as 2D Plots	6-5

Contents-1

Exporting Total Radiated Power	6-6
7 - Frequency Sweep of Voltages	7-1
Disabling Voltage Sources	7-1
Creating a Current Source on a Component	7-1
Creating a Voltage Probe	7-4
Running a Frequency Sweep	7-6
Plotting Probe Voltage	7-7

# 1 - Introduction

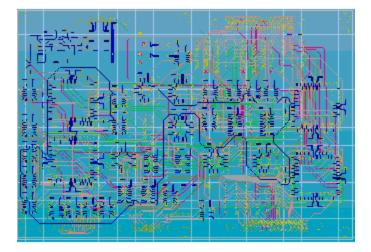
This Getting Started Guide is intended to quickly familiarize you with the capabilities of Slwave. This guide leads you step-by-step through importing a PCB design, setting up and performing three Slwave analyses, and viewing the results of the simulations.

This guide explains how to perform the following tasks in Slwave:

- Importing a geometric PCB model
- Validating the design
- Running three Slwave simulations:
  - Resonant Modes Analysis
  - SYZ Analysis
  - PSI SYZ Analysis
  - AC Current Analysis
  - Frequency Sweep of Voltages
- Adding ports, sources, and probes to the PCB as appropriate for each analysis
- Specifying parameter settings for each type of analysis
- Creating 2D plots of the results, and comparing the results from the three solutions

## **The PCB Model**

The PCB model used in this Getting Started Guide consists of an 8-layer PCB. Layers 2 and 7 are planes for power and ground.



Introduction 1-1 Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

## **Expected Results**

The analyses will demonstrate how frequency-dependent impedance and voltage differences between the power and ground planes can cause signal integrity issues such as voltage ripple. The resonant modes identified in the Resonant Modes Analysis can be clearly seen in the impedance plots from the SYZ Analysis and the graphs of voltage swings from the Frequency Sweep.

# 2 - Setting Up the Design

This section explains how to perform the following tasks:

- Importing and saving a project
- Setting the visibility of geometric and circuit elements
- Viewing the Layers workspace and layer stackup
- Identifying power/ground nets
- Running a validation check on the design

## Importing and Saving the Project

To begin, import the PCB design and its components from an Ansys Neutral File (\*.anf). This file contains information about the PCB's geometry, layer stackup, padstacks, vias, materials, and discrete components.

- 1. Launch Slwave.
- 2. If the Welcome to Slwave window opens at launch, close it.
- 3. Click the Import tab.
- 4. In the Ansys EDA Layouts area, click ANF.

The Select Ansoft Neutral File to Import window appears.

- 5. Depending on your operating system, navigate to one of the following locations:
  - Windows: \Program Files\AnsysEM\v222\Win64\Examples\Slwave
  - Linux: /Program Files/AnsysEM/v222/Linx64/Examples/Slwave
- 6. Select the file **siwave\_board.anf**.
- 7. Click Open.

The Select nets to import from siwave\_board window appears.

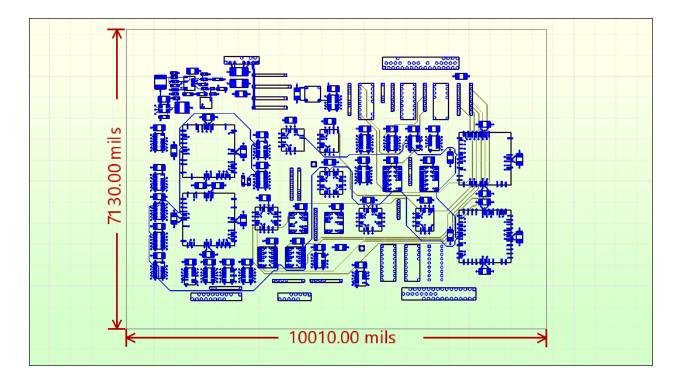
ad Net Configuration File			Browse	Load
Net Name	Import	Setup Ports	Port Reference Net	
ABC	$\checkmark$		GND	
BLT_DATA_P1	$\checkmark$		GND	
BLT_DATA_P2	$\checkmark$		GND	
BLT_DATA_P3	$\checkmark$		GND	
BLT_DATA_P4	$\checkmark$		GND	
BLT_DATA_R1	$\checkmark$		GND	
BLT_DATA_R2	$\checkmark$		GND	
BLT_DATA_R3			GND	
BLT_DATA_R4			GND	
CLK_1K			GND	
CLK_125K			GND	
CLK_156K			GND	
CLK_312K CMD_EXECUTE			GND	
	o Import: 308		GND	
ld card: Se	ect Matching Rows De	select Matching R	ows	
Operations				
Import? Yes V Update				
Setup Ports? Yes V Update	Port Reference Na	st: GND	V Upda	ate

8. Leave the settings as-is, and click **Import Configuration**.

If a Component Import Overwrite message appears, click **Yes to All** to overwrite any existing names.

If the Slwave Workflow Wizard opens, close it to reveal the Modeling workspace.

The design should look like this:



Note:	
You can change S	Iwave's background colors from the <b>View</b> tab:
Selection Color Background Stighting	

9. Click File > Save As.

The Save As window appears.

- 10. Browse to a directory where you have write permission. Enter a name for this project, such as **siwave\_board\_test1.siw**.
- 11. Click Save.

# **Setting PCB Element Visibility**

PCB element visibility is controlled from the Layers workspace and the View tab.

To turn on visibility for geometric elements (planes, traces, pads, vias, and circuit elements):

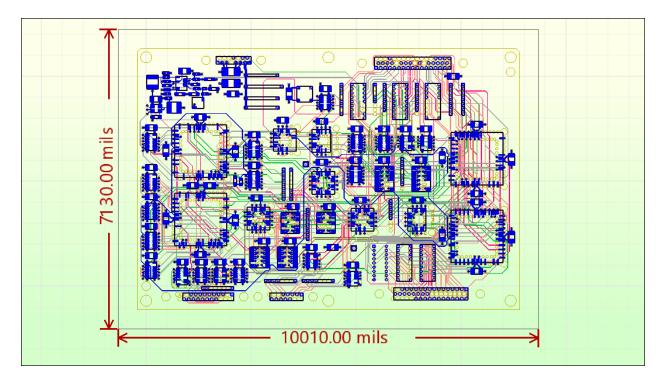
Setting Up the Design 2-3

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

- 1. Navigate to the Layers workspace.
- Use the Show All check boxes (■) to enable visibility for all parts on every layer.
   When you are finished, the Layers workspace should look like the following:

Layers							<b>–</b> 1	ч×
Show Dielec	trics							
		×	0	00	۲	۵	Ť	
SURFACE		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OL2		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OL3		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OL4		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OL5		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
○ L6		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OL7		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	
OBASE		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	

The **Modeling** workspace should look like the following:



Setting Up the Design 2-4

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

#### Note:

While no further changes are necessary for this Getting Started Guide, additional visibility options are described in Slwave's online help.

## Viewing the Layers Workspace and Layer Stackup

The **Layers** workspace and the **Layer Stackup Editor** control the visibility and properties of the package layers.

Layers can be viewed in either outline or filled mode.

To set the layers for this project to filled mode:

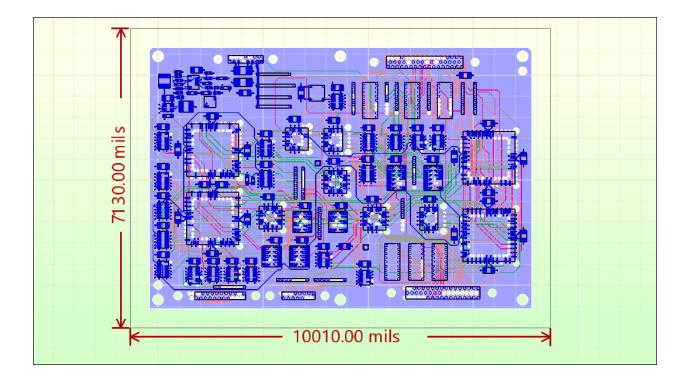
- 1. Navigate to the Layers workspace.
- 2. Click within the colored rectangles to change layers from outline to solid fill:
  - Set Surface, L7, and BASE to solid fill.
  - Leave the remaining layers as outlines.

The Layers workspace should look like the following:

Layers						,	•	Ŧ	×
Show Dielec	trics								
		×	0	00	۲	۵	Ť		
SURFACE		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
OL2		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
OL3		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
OL4		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
OL5		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
○ L6		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
OL7		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		
BASE		X	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$	$\checkmark$		

The Modeling workspace should look like the following:

Setting Up the Design 2-5



## Viewing the Layer Stackup

To view the layer stack:

- 1. Navigate to the **Home** tab.

The Layer Stackup Editor appears.

Color	Name	Туре	🔶 🕂 Th	ickness (mils)	Aterial Material	Conductivity (S/m)	Ca Dielectric Fill	Dielectric constant	Loss tangent	Translucency	Elevation (mils)	Roughness (mils)	Trace Cross-section	
	UNNAMED_1	DIELECTRIC	0		air	0		1.0006	0		52.08			
	SURFACE	METAL	0.72		copper	5.8E+07	air	1.0006	0	0	51.36	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	1
	UNNAMED_3	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		45.36			
	L2	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	43.92	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	1
	UNNAMED_5	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		37.92			
	L3	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	36.48	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	
	UNNAMED_7	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		30.48			
	L4	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	29.04	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	
	UNNAMED_9	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		23.04			
	L5	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	21.6	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	1
	UNNAMED_11	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		15.6			
	L6	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	14.16	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	1
	UNNAMED_13	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		8.16			
	L7	METAL	1.44		copper	5.8E+07	FR4_epoxy	4.4	0.02	0	6.72	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	_
	UNNAMED_15	DIELECTRIC	6		FR4_epoxy	0		4.4	0.02		0.72			١,
	BASE	METAL	0.72		copper	5.8E+07	air	1.0006	0	0	0	HJ: 0 , HJ: 0 , HJ: 0	Rectangle	_
	ete / Move Layer	(s) E	dit Selec	ted Layer(s)					-					
Add A	Above Selected L	ayer	Color	0000ff		Update	Dielectric Fill	air	~	Update				
Add I	Below Selected La							•					1	
Del	ete Selected Lav	erc N	lame	SURFACE		Update	Translucency		0%	Update				
			ype	METAL	~	Update	Thickness	0.72	mils	Update	10000	10000	<b>\$</b>	
Move	e Selected Layers	s Up	Abc.									-		
Move	Selected Layers I	Down N	1aterial	copper	~	Update	Roughness	HJ: 0 , HJ: 0 , HJ: 0	) mils	Update				

Setting Up the Design 2-6

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

3. Here, you can change the properties for one or more layers. For now, you need only be aware that this window exists. Later, this guide will focus on the interactions between layers L2 and L7.

## **Identifying Power and Ground Nets**

In this section, you will explore the **Nets** workspace.

#### Note:

By default, the **Nets** workspace is located on the upper-left side of the Slwave window. Workspaces can be moved by dragging and dropping them to another location.

Before performing any simulations, power and ground nets in the design must be identified.

Nets containing large planes must be classified as Power/Ground nets. Signal nets containing microstrip and stripline routing need to be classified as Non Power/Ground nets. This enables the solver to judiciously choose the mesh refinement and optimization strategies for the signal and power/ground nets.

To identify power/ground nets:

1. In the **Nets** workspace, which defaults to **Single Ended Nets**, select the **Power/Ground Identification** tab.

Power/Ground Identification 👻	Ψ×
Regular Exp:	
Non Power/Ground Nets	
ABC BLT_DATA_P1 BLT_DATA_P2 BLT_DATA_P3	<
Power/Ground Nets	
GND P28VA VCC	
Auto Identify	
🕂 Single 🛟 Differe 🗧 Extend 🛬 Pov	wer//

Setting Up the Design 2-7

2. Click Auto Identify to have Slwave automatically classify the power and ground nets.

Power/Ground Identification $\bullet$ # ×
Regular Exp:
Non Power/Ground Nets
ABC  BLT_DATA_P1 BLT_DATA_P2 BLT_DATA_P3  V
Power/Ground Nets
GND P28VA VCC
Auto Identify
Fomeren Conteren Power/

Nets GND, P28VA, and VCC should be classified as Power/Ground Nets.

3. If any net is incorrectly identified, click to highlight the net name and use the up and down arrows to move it to the correct list.

# **Running a Validation Check**

Run a validation check of the PCB design before running any simulations. This check identifies several common layout and design errors.

1. Click Tools. In the Inspection Tools area, click Validation Check.

The Launch Validation Check window appears.

Setting Up the Design 2-8

Launch Validation Check	×
Check List Select All Unselect All	Select a simulation mode No Associated Simulation
Self-Intersecting Polygons	Strict Disjoint Net Checking
Disjoint Nets (Floating Nodes)	Minimum Area: 3100.01 mils^2
DC-Short Errors	Cutouts that are smaller than this minimum area will be ignored during validation check.
Identical/Overlapping Vias	This threshold can be changed in the
Bondwire Collisions	Simulation -> Global Option window.
Illegal Bondwire Connections	Nets to be checked
Misalignments	Some nets might not be included. Please refer the Simulation -> Global Option
Less Than Two Terminals	window.
	Number of cores to use: 4
ОК	Cancel

2. Leave the default settings, and click **OK** to start the check.

The **Messages** window updates with a **Process Monitor** showing the status of the validation check.

Process Monitor (Validation Checker)	<b>→</b> ∓ ×
Display: Messages 👻 🖬 🕨	
Checking Disjoint Net: TCI4_LS Checking Disjoint Net: TCI4_DIG3_PR_SWITCH Checking Disjoint Net: TCI4_DIG3_ON_SWITCH	^
Checking Disjoint Net: TCI4_DIG3_ON	~
<	>
Progress: 09%	
Messages Process Monitor (Validation Checker)	

When the process is finished, the Validation Check Results window appears.

Setting Up the Design 2-9

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

Validation Check Results	×
Errors	
Self-Intersecting Polygons: 0	Auto Fix
Point-Connections:	
Disjoint Nets: 0	Auto Fix
DC-shorted Errors: 0	Auto Fix
Identical/Overlapping Vias: 0	Auto Fix
Traces-Inside-Traces Errors: 0	Auto Fix
Collisions of Bondwires: 0	
Illegal Connections of Bondwires: 0	
Identical Bondwires: 0	Auto Fix
Reversed Bondwires: 0	Auto Fix
Floating Nodes: 0	Auto Fix
Zero Via Plating: 0	
Nets With Less Than 2 Terminals: 0	
Warnings	
Misalignments (Planes/Traces/Vias): 0	Auto Fix
Bondwires Misaligned With Die Pads: 0	Auto Fix
Pins Shared By Multiple Pin Groups: 0	
Self-Intersection Warnings: 0	
Components With Pins From Multiple Layers: 0	Auto Fix
OK	Cancel
UK	Cancel

If there were errors or warnings that Slwave could resolve, the **Auto Fix** check boxes would be available.

This design shows no errors, and is ready for simulation.

- 3. Click **OK** to close the window.
- 4. Click **FILE > Save** to save the updated PCB design.

Setting Up the Design 2-10

# **3 - Resonant Modes Analysis**

This section explains how to perform the following tasks:

- Running a Resonant Modes analysis
- Viewing the results as a data table or as 2D plots

### **Running the Resonant Modes Analysis**

Traces routed through power and ground planes can exhibit signal integrity problems. The resonant mode calculation is the first step in identifying non-ideal plane behavior that can affect signal integrity.

Set up an Slwave Resonant Modes analysis:

1. Click Simulation > Compute Resonant Modes.

The Compute Resonant Modes window appears.

- 2. Verify that **Minimum Frequency** is set to 2.55238E+08. If it is not, click **Restore Recommended Minimum Frequency**.
- 3. Set the Maximum Frequency to 2e9 (2 GHz).
- 4. Set the # of Modes to compute to 10.

Compute Resonant Modes	×
Simulation name:	
Resonant Mode Sim 1	-
Find Modes in Frequency Range	
Minimum Frequency: 2.55238E+08 Hz	
Restore Recommended Minimum Frequency	
Maximum Frequency: 2e9 Hz	
# of Modes to Compute: 10	
Other solver options	
Save Settings Launch Close	

Resonant Modes Analysis 3-1

5. Click Launch to begin the analysis.

The **Messages** workspace updates with a progress bar.

When the analysis has finished, it appears in the **Results** workspace:

Results	•	д	×
Resonant Modes			

### **Viewing Resonant Modes Analysis Results**

After the analysis has finished, Resonant Modes results can be viewed in a table, or as twodimensional plots overlaid on the PCB design.

To view tabular results:

1. In the **Results** workspace, double-click **Resonant Mode Sim 1** (or whatever you've named the simulation).

The Resonant Modes window appears.

Resonant Modes Analysis 3-2

-	nant Modes(Reson	ant Mode Sim 1)			_ 🗆	×
Mode	Re. Freq (GHz)	Im. Freq (GHz)	k	Wavelength (m)	Q	1
1	0.289528107	0.002907360	6.06806041	1.035452003	49.79478810	0
2	0.325361938	0.004624167	6.81908197	0.921412198	35.18415640	0
3	0.431892277	0.004363999	9.05179278	0.694137113	49.48607550	0
4	0.526022548	0.005364308	11.02461739	0.569923208	49.03241070	0
5	0.587586516	0.005934639	12.31490274	0.510209901	49.50734500	0
6	0.648920180	0.009212841	13.60036130	0.461986646	35.22179470	0
7	0.741151551	0.007580190	15.53338788	0.404495488	48.88995830	0 .
C						>
	DL + L		Reference Layer			
Mode	Plot Layer					
Mode	Plot Layer					
Mode	Plot Layer					
Mode	Plot Layer					
Mode	Plot Layer					
Mode	Plot Layer					
Mode	Plot Layer					

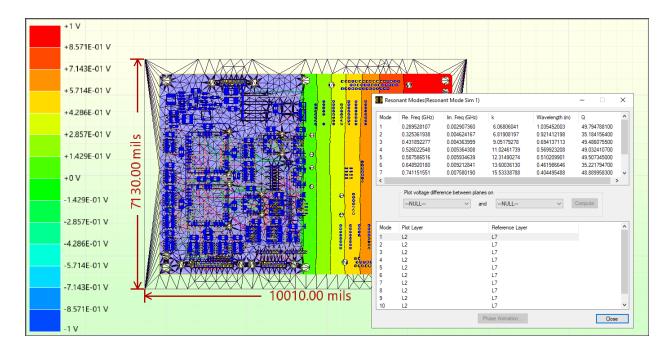
At the center of this window, two drop-down menus allow you to compute the voltage differences between two planes.

Perform the following steps to plot the voltage differences between layers L2 and L7, the power and ground planes:

- 1. Use the drop-down menus under **Plot voltage difference between planes on** to select **L2** and **L7** respectively.
- 2. Click **Compute** to generate the 2D plot data from the solution data.

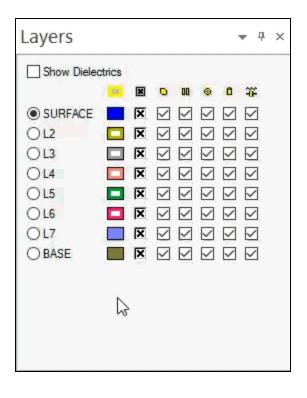
The lower part of the window updates to display the modes available for plotting. There are ten modes, and the underlying Modeling workspace updates to display Mode 1.

Resonant Modes Analysis 3-3

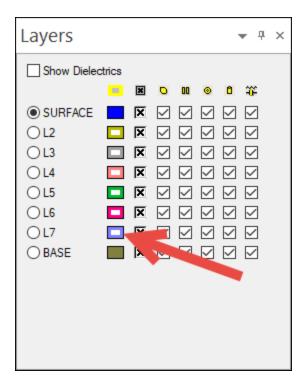


You'll notice that layer L7's display mode obscures part of the plot.

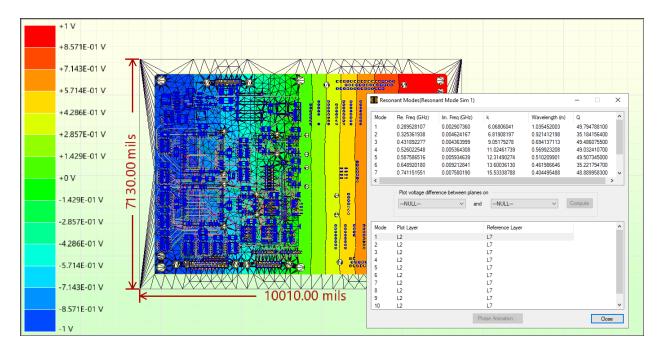
3. From the Layers workspace, click the solid color block next to L7 to revert it to outline mode.



Resonant Modes Analysis 3-4



The entire plot is now visible:



4. Click on each row in turn to see the voltage difference as a color map on the PCB. As indicated on the left, areas with large positive voltages appear as red, while areas with large negative voltages appear as blue.

Resonant Modes Analysis 3-5

5. To view the modes in sequential animation, click **Phase Animation**.

The **Phase Animation** window appears.

Phase Animation		_	×
Re. Freq: 0.289528       (Gi         Im. Freq: 0.00290736       (Gi         Start       0       degre         End Angle:       360       degre         Step Size:       20       degre	iz) ees ees		
Total number of frames: 18			
Generate Frames	1		
Evport Close			
Export Close			

6. Leave the **Start**, **End Angle**, and **Step Size** settings as their defaults, and click **Generate Frames**.

The **Frames** area updates with a list.

	Phase =	
Frame 1;	Phase =	20.00
Frame 2;	Phase =	40.00
Frame 3;	Phase =	60.00
	Phase =	
	); Phase =	
	1; Phase =	
	2; Phase =	
	3; Phase =	
	1; Phase =	
	5; Phase =	
	5; Phase =	
	7; Phase =	
Frame 18	3; Phase =	= 360.00

Resonant Modes Analysis 3-6

- 7. Click through the frames one by one to change the underlying Modeling workspace to that frame, or use the **Play** and **Pause** buttons to control an animation of the frames.
- 8. Click **Close** to terminate the animation.
- 9. Click **Close** again to close the **Results** window.
- 10. Click **File > Save** to save the design with your Resonant Modes analysis results.

Resonant Modes Analysis 3-7

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF layout 3-8

# 4 - Slwave SYZ Analysis

This section explains how to perform the following tasks:

- Defining pin groups
- Defining a port between the pin groups
- · Generating the SYZ-parameters for selected power supply nets
- Viewing the frequency-dependent impedance of the planes on an X-Y plot

## **Defining Pin Groups for GND and VCC**

To launch the Pin Group Manager:

1. Click Tools > Create/Manage Pin Groups.

The Create/Manage Pin Groups window appears.

Create/Manage Pin Groups	×
Create/Manage Pin Groups  Part Name:  Reference Designator:  Nets List common nets only Show Pin Numbers Show Pin Names  Net Selection Select all nets Unselect all nets Unselect all nets Hide All Nets Hide All Nets	Create Pin Group(s) Naming Convention
	Delete Pin Group(s)     Create Port       Edit Pin Group     Create Terminal
	Close

Define a pin group for the **GND** net:

Slwave SYZ Analysis 4-1

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

- 1. From the **Part Name** menu, select **MACH230\_SMSOCKETAMD**.
- 2. From the Reference Designator list, select U41.
- 3. In the **Nets** list, ensure that the check box next to **GND** is selected, and that no other nets are selected.
- 4. Click Create Pin Group(s).

The Pin Group appears in the Pin Group List.

Define a pin group for the **VCC** net:

- 1. In the **Nets** list, deselect the check box next to **GND** and select the check box next to **VCC**. Ensure that no other check boxes are selected.
- 2. Click Create Pin Group(s).

Both the GND and VCC pin groups should now appear in the Pin Group List.

Create/Manage Pin Groups	×
Part Name:	Options
MACH230_SMSOCKETAMD V	Create pin groups for each part
	Create pin groups for each net
Reference Designator:	Create pin groups per grid cell
U14 U22	Row # Col #
022	
U41	Synchronize grid lines' movement
	Delete existing pin groups
No. 4	Pin Group List
Nets	U41_GND_Group
List common nets only Show Pin Numbers Show Pin Names	U41_VCC_Group
GND	
✓ vcc	
CLK_1K Select all nets	
CLK_125K Unselect all nets	
FORCEOFF	
HLC_ERR ONet Visibility	
HLC_ISOLATE Show All Nets	
	Create Pin Group(s) Naming Convention
HLC_RRC_RDY Hide All Nets	
HLC_RRC_RESET	Delete Pin Group(s) Create Port
	Edit Pin Group Create Terminal
	Close

3. Click Close.

# **Defining a Port Between Pin Groups**

Define a port on component U41.

- Click Tools > Generate Circuit Element on Components. The Circuit Element Generation Dialog appears.
- 2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230\_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
- 3. In the **Reference Terminal Component** section, enable the **Same as Positive Terminal** check box, which will populate the Reference Terminal Component fields with the same information.
- 4. In the **Circuit Element Positive Terminal** section, expand the **Pin Groups** list and select the **U41\_VCC\_Group** pin group.
- 5. In the **Circuit Element Reference Terminal** section, expand the **Pin Groups** list and select the **U41\_GND\_Group** pin group.
- In the Circuit Element Type section, select the Port radio button.
   Your selections should look like the following:

Circuit Elen	nent Generation Dialog					-		×
Positive Termin	nal Component		Reference T	erminal Comp	onent			
Part Name:	MACH230_SMSOCKETAMD	~	Part Name:	MACH230_9	SMSOCKETAMD			$\sim$
Ref Des:	U41	~	Ref Des:	U41				$\sim$
			Same as I	Positive Termi	inal			
Circuit Element	t Positive Terminal	Circuit Element Reference	Terminal		Circuit Elements			
CLK_1K     CLK_1K     CLK_1Z     FORCEC     GND     HLC_AN     HLC_CT     HLC_CT     HLC_ER     HLC_CR     HLC_CR     HLC_RR     HLC_RR     HLC_RR     HLC_RR     HLC_RR     HLC_ST     HLC_ST     HLC_ST     HLC_ST     TC11_DI     TC11_PC     TC11_PC     TC12_DI     TC12_DI	141_GND_Group 141_VCC_Group 55K OFF VAILABLE OMPLETE TL RR OLATE DADSHED RC_RDY RC_RESET 5T_N HIFT FART TOP 5 N_PRI	Pin Groups Pin Group State Pin Gro	Find Pin at Lo ference pin reference distar O Current O Voltage	ice Source Source	Capacitors Current Sources Inductors Resistors Ports Voltage Probes Voltage Sources Terminals		Edit	
		Naming Convention	Create			ОК	Car	ncel

7. Click Create.

The Port Properties window appears.

Slwave SYZ Analysis 4-4

Port Properties		$\times$
Name:	U41_VCC	
Reference Impedance:	1  Ohms	
Positive Terminal Net: V0	00	
Negative Terminal Net: 6	âND	
Positive Terminal Pingrou	ip: U41_VCC_Group	
Reference Terminal Ping	roup: U41_GND_Group	
ОК	Cancel	

- 8. Shorten the name to **U41\_VCC**.
- 9. Set the **Reference Impedance** to **1** Ohm.
- 10. Click **OK** to accept the port definition.
- 11. Click **OK** to exit the **Circuit Element Generation** window.

### **Generating SYZ Parameters**

Next, we calculate the frequency-dependent impedance response at port U41\_VCC.

The active components on the PCB draw current through the power supply nets, such as VCC. If the impedance of the VCC net is too large, ripple voltage may be induced between VCC and GND when the components switch. The frequencies previously identified by the Resonant Modes Analysis correspond to peaks in the impedance of the power supply VCC net.

1. Select the Simulation tab. In the Slwave area, click Compute SYZ Parameters.

#### Note:

Ensure you are in the correct area of the **Simulation** tab, as there is another **Compute SYZ Parameters** link for PSI.

The **Compute SYZ-parameters** window appears, populated with default settings.

Slwave SYZ Analysis 4-5

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

Compute SYZ-p	aramet	ers					×
Sweep Sensitiv	ity Di	stributed Analy	sis (HPC)				
Simulation n	ame:	SYZ Sweep 1					
_	L						
		DC point					
Frequency F			N	· · · · / C · · · · · ·	Distri		_
1 SMHz	Freq	Stop Freq 5GHz	100	oints / Step Size	Linear	bution	_
Add	Above	Add E	elow	Delete Select	tion	Preview	
S	ave	Loa	ad	Set Defaul	lt	Clear Default	
Sweep Selec				]Set FWS genera	ation pa	arameters	
		veep		Min Rise/Fall T	īme / s		
		or S: 0.5 %	,	1E-10			
				3D Solver			
Passivity/Ca	ausality			Q3D (auto-d			
Enforce	e Causa	-		HFSS (user-o		f <b>regions)</b> hematic (do not si	imulate)
	e Passiv	nty		Solve reg		-	gure
				HFSS solv			guierri
Export Tou	chstone	® file after sin	ulation c	ompletes		Other solver	options
File path:	eDrive	e - ANSYS, Inc/	Desktop/	siwave_boardGS	G.s1p	E	rowse
				Save Settin	ngs	Launch	Close

- 2. In the Frequency Range Setup section, set the Start Freq to 0 Mhz.
- 3. Set the **Stop Freq** to **1 GHz**. Note that the **Min Rise/Fall Time** value changes to fit the maximum frequency (in this case, 5e-10).
- 4. Set the Num. Points to 200.
- 5. Leave the **Distribution** as **Linear**.

Slwave SYZ Analysis 4-6

- 6. In the Sweep Selection section area, ensure that Discrete Sweep is selected.
- 7. Leave any other settings as-is.

Your settings should look like the following:

Compute SYZ-parameters	>
Sweep Sensitivity Distributed Analysis (	(HPC)
Simulation name: SYZ Sweep 1	~
Compute exact DC point	
Frequency Range Setup	
	Im. Points / Step Size Distribution
1 OMHz 1GHz 200	0 Linear
Add Above Add Below	w Delete Selection Preview
Save Load	Set Default Clear Default
<ul> <li>Discrete Sweep</li> <li>Interpolating Sweep</li> <li>Relative error for S: 0.5 %</li> </ul>	Set FWS generation parameters Min Rise/Fall Time / s 5E-10 3D Solver
Passivity/Causality	Q3D (auto-detected regions)
Enforce Causality	HFSS (user-defined regions)
Enforce Passivity	AEDT regions schematic (do not simulate) Solve regions in parallel Configure
	HFSS solver options
Export Touchstone® file after simulat	tion completes Other solver options
File path: eDrive - ANSYS, Inc/Des	ktop/siwave_boardGSG.s1p Browse
	Save Settings Launch Close

8. Click Launch to launch the SYZ analysis.

Slwave SYZ Analysis 4-7

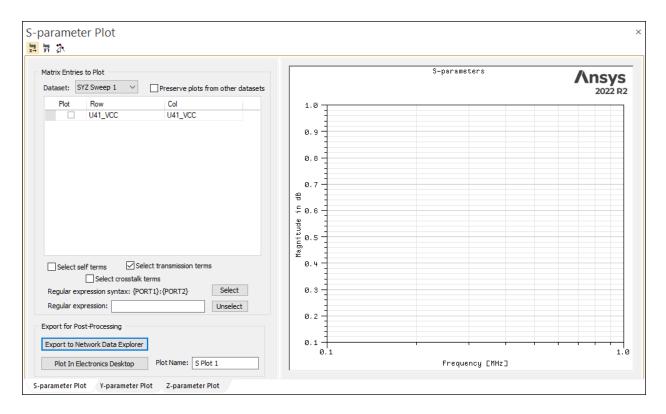
The **Messages** workspace updates with a progress bar showing simulation progress. When it has finished, you can view the impedance response.

### Viewing Impedance Response

To view the frequency-dependent impedance response:

 Select the Results tab. In the Slwave area, click the SYZ icon to access its drop-down menu. Select SYZ Sweep 1 > Plot Magnitude....

The S-parameter Plot window opens.



Click the **Z-parameter Plot** tab at the bottom of the window.
 The window changes to **Z-parameter Plot**.

Slwave SYZ Analysis 4-8

atrix Entries to Plot ataset: SYZ Sweep 1  Preserve plots from other datasets		Z-parameters	<b>Ansys</b> 2022 R
Plot         Row         Col           Image: U41_VCC         U41_VCC	1.00E+10	Z(U41_VCC,U41_V SYZ Sweep 1 Z Plot	(00)
V 041_VCC 041_VCC	1.00E+09		
	1.00E+08 -		
	1.00E+07		
	1.00E+06		
	9 1.00E+05		
	a) 1.00E+05 1.00E+04 E 1.00E+04 <sup>™</sup> 1.00E+03 <sup>™</sup> 1.00E+03 <sup>™</sup>		
	∰ 1.00E+03		
Select self terms	1.00E+02		
Select crosstalk terms Regular expression syntax: /PORT1} Select	1.00E+01 -		A . 4
Regular expression syntax: {PORT1}:{PORT2}         Select           Regular expression:         Unselect	1.00E+00		
port for Post-Processing	1.00E-01		/ 1

Your port should automatically be selected for plotting. If it is not, enable the check box in the **Plot** column.

#### Note:

This design only has one port defined, but it is possible to display multiple plots at once.

#### Note:

You can change many aspects of the plot, including labels, scaling, and color. Double-click anywhere within the plot window to launch the **Properties** window.

- 3. Close the report.
- 4. Click FILE > Save to save the project with the impedance result.

Slwave SYZ Analysis 4-9

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

PDF layout 4-10

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

# 5 - PSI SYZ Analysis

This section explains how to perform the following tasks:

- Generating SYZ parameters for selected power supply nets using the PSI solver
- Viewing the frequency-dependent impedance of the planes on an X-Y plot

#### Prerequisite: Defining Pin Groups and Ports

The PSI solver uses the same pin groups and port settings as the SIwave SYZ analysis.

If you have not yet completed the previous chapter (<u>Slwave SYZ Analysis</u>), please do so to set up your pin groups and ports.

### **Generating SYZ Parameters using PSI**

Next, calculate the frequency-dependent impedance response at port U41\_VCC using the 3D full-wave PSI solver.

1. Select the Simulation tab. In the PSI area, click Options.

Note:

Ensure that you click the PSI **Options** icon, as there are others.

The PSI Options window appears.

il Options		;
eneral Net Processing Power/Ground Nets Signal Nets Exte	rnal Environment	
<ul> <li>Local analysis (solve on local machine or single remote server)</li> <li>Number of cores to use: 32</li> <li>HPC License Type</li> <li>Pool</li> <li>Pack</li> <li>Remote server name: localhost</li> <li>Port: 31000</li> <li>Distributed analysis (HPC on multiple servers)</li> <li>Configure</li> <li>Simulation Preference</li> </ul>	Model Type       RMS surface roughness:       PCB         Ignore geometry smaller than       5660.2mil2         Ignore voids smaller than       3.3124mil2         Snap vertices separated by less than       0.0984252mil         Enhanced bond wire modeling       Conductor Surface Roughness         Model:       None         RMS surface roughness:       0	Restore Default Restore Default Restore Default
Balanced Accuracy		
Temporary working folder:		
<pre><li><li><li><li></li></li></li></li></pre> // <pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//<pre>//&lt;</pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre></pre>	Browse	
Perform ERC during simulation setup		
xport Settings Import Settings		DK Cancel

2. Ensure that **Local Analysis** is selected. In the box below, use the up and down arrows to change the **number of cores to use**. To use all cores, click the up arrow until you reach the maximum.

For best performance, Ansys generally recommends at least 8, and optimally 12- 16 cores.	Important:

- 3. Leave all other options as-is, and click **OK** to exit the **PSI Options** window.
- Navigate to the Simulation tab. In the PSI section, click Compute SYZ Parameters. The Compute SYZ-parameters using PSI window opens.

PSI SYZ Analysis 5-2

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

Compute	e SYZ-parame	ters using PSI						Х
Sweep								
Simulation name: PSI S-parameter Sweep ~								
	Start Freq Stop Freq Num. Points / Step Size Distribution			ution				
1	1 10MHz 2GHz 100 Linear							
	Add Above	Add B	elow	Delete Select	ion	Preview		
[	Save	Loa	ad	Set Defaul	t	Clear Default		
0	ep Options Discrete Sweep Interpolating Sv			Set FWS genera Min Rise/Fall T 2.5E-10		ameters		
]	Fast Sweep Adaptive Sar Enforce D and causa	m <b>pling</b> IC point		Set por	t type			
Export Touchstone® file after simulation completes File path: eDrive - ANSYS, Inc/Desktop/siwave_boardGSG.s1p Browse								
rite	e path: eDriv	e - ANSTS, INC/	Desktop/s	wave_poard65	dis th		Browse	
				Save Settin	ngs	Launch	Close	

- 5. Ensure the Start Freq is set to 10MHz.
- 6. Set the Stop Freq to 1GHz.
- 7. Under Sweep Options, select Interpolating Sweep (AFS), and select Fast Sweep and Adaptive Sampling. The Fast Sweep algorithm improves run time by requiring fewer samples to simulate, but uses more memory. Fast Sweep may not be used if insufficient

PSI SYZ Analysis 5-3

memory is available.

Your settings should look like the following:

Compute SYZ-paramete	ers using PSI					×	
Sweep							
Simulation name:	PSI S-paramete	er Sweep				~	
Start Freq	Stop Freq	Num. Po	ints / Step Size	Distrib	oution		
1 10MHz	1GHz	100		Linear			
Add Above	Add B	elow	Delete Select	tion	Preview		
Save	Loa	d	Set Defau	lt	Clear Default		
Sweep Options			Set FWS genera Min Rise/Fall T 5E-10				
<ul> <li>Interpolating Swe</li> <li>Fast Sweep</li> <li>Adaptive Sam</li> <li>Enforce DC and causali</li> </ul>	pling Cpoint		Set por	t type.			
Export Touchstone® file after simulation completes File path: eDrive - ANSYS, Inc/Desktop/siwave_boardGSG.s1p Browse							
			Save Settin	ngs	Launch	Close	

PSI SYZ Analysis 5-4

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

8. Click Launch to begin the PSI SYZ analysis.

The **Messages** workspace updates with a progress bar. When the simulation has finished, you can view the impedance response.

### **Viewing Impedance Response**

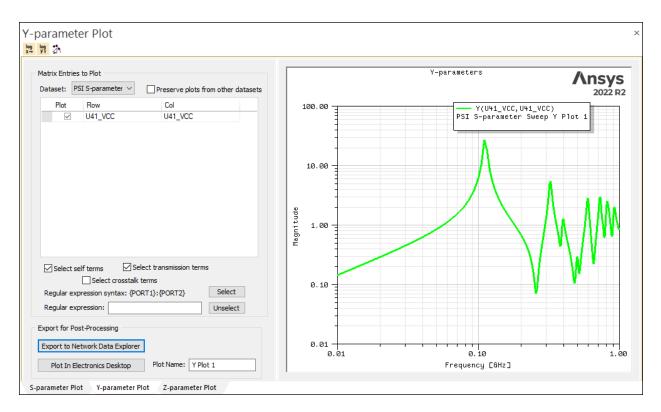
To view the frequency-dependent impedance response:

1. From the Results workspace, double-click PSI S-parameter Sweep.

#### Note:

If you have hidden your **Results** window, navigate to the results using the **Results** tab. In the PSI area, click **SYZ** > **PSI S-Parameter Sweep** > **Plot Magnitude**.

The **Y-parameter Plot** window opens.



PSI SYZ Analysis 5-5

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

atrix Entries to Plot Dataset: PSI S-parameter V Preserve plots from other datasets		Z-parameters	
Plot         Row         Col           Image: Col         U41_VCC         U41_VCC	100.00	Z(U41_VCC,U41_ PSI S-parameter Swe	VCC) ep Z Plot 1
	10.00		Λ.
	υ υ υ υ υ υ υ υ υ υ υ υ υ υ		$\mathbb{N}$
Select self terms Select transmission terms Select crosstalk terms Regular expression syntax: {PORT1}: {PORT2} Regular expression: Unselect	0.10		
Export for Post-Processing Export to Network Data Explorer Plot In Electronics Desktop Plot Name: Z Plot 1	0.01	0.10 Frequency [GHz]	1.

2. Click the **Z-parameter Plot** tab to view the plot.

3. Click **FILE > Save** to save the project with the PSI SYZ results.

PSI SYZ Analysis 5-6

## 6 - PSI AC Current Analysis

This section explains how to perform the following tasks:

- Creating a Voltage Source
- Calculating AC currents on selected power supply nets
- Viewing current distributions as 2D plots
- Exporting total radiated power

### **Creating a Voltage Source**

AC Current analysis requires the existence of one or more sources as excitations.

Because the analysis result is a current, the effect of the changing plane impedance will be most clear if a constant voltage excitation is used.

To create a voltage source:

1. Click Tools > Generate Circuit Element on Components.

The Circuit Element Generation window appears.

- 2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230\_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
- 3. In the **Reference Terminal Component** section, check the **Same as Positive Terminal** check box to select the same.
- 4. In the Circuit Element Positive Terminal section, expand the Pin Groups option and select U41\_VCC\_Group.
- 5. In the **Circuit Element Reference Terminal** section, expand the **Pin Groups** option and select **U41\_GND\_Group**.
- 6. In the Circuit Element Type section, select Voltage Source.

Your settings should look like the following:

PSI AC Current Analysis 6-1

Positive Terminal	Component							
	component			Reference T	erminal Comp	onent		
Part Name: N	MACH230_SMSOCKETA	MD	~	Part Name:	MACH230_	SMSOCKETAMD		
Ref Des: L	U41		~	Ref Des:	U41			
				Same as	Positive Termi	inal		
Circuit Element Po	ositive Terminal		Circuit Element Reference	Terminal		Circuit Elements		
CLK_1K CLK_1Z5K FORCEOFI GND HLC_AVAI HLC_CON HLC_CTL HLC_CCN HLC_CCN HLC_SOL HLC_SOL HLC_RR HLC_SOL HLC_RRC HLC_SOL HLC_SOL HLC_SOL HLC_SOL TCI1_ON TCI1_ON TCI1_POW TCI1_POW TCI2_DIG1 Expand Colla	I_GND_Group _VCC_Group SF ILABLE MPLETE ATE _ATE _DSHED _RDY _RESET _N T RT P PRI RED VER_AVAILABLE		Pin Groups     Pin Group	Find Pin at Lo	nce Source Source	Capacitors Current Sources Resistors Ports Voltage Probes Voltage Sources Terminals		
Reg exp:	X		O Terminal	Voltage	FIODE	Delete	Edit	

7. Click Create.

The Set Voltage Source Properties window appears.

PSI AC Current Analysis 6-2

Set \	/oltage Source Properties			×				
Nam	ne: U41_VCC_VSRQ							
• F	Frequency Independent Parameters							
	Magnitude: Parasitic Resistance:	1 1E-06	Volts Ohms					
	Phase:	0	Degrees					
O Frequency Dependent								
Path: Browse File should contain data in <freq> <real> <imag> format</imag></real></freq>								
		[	OK	Cancel				

- 8. Shorten the name to U41\_VCC\_VSRC.
- 9. Ensure that **Frequency Independent** is selected.
- 10. Ensure the **Magnitude** is set to **1 Volt**, the **Parasitic Resistance** is set to **1E-06 Ohms**, and the Phase is set to **0 Degrees**.
- 11. Click **OK** to accept the Voltage Source definition.
- 12. Click **OK** to exit the **Circuit Element Generation** window.

### **Calculating AC Currents**

The AC Currents simulation calculates surface current, and can reveal regions of high impedance that may occur due to antipads, cutouts, or discontinuities in ground planes.

To calculate the surface current flowing on the VCC and GND planes:

1. Select the **Simulation** tab. In the **PSI** area, click **Compute AC Currents**.

The Compute AC currents using PSI window appears.

PSI AC Current Analysis 6-3

Compute AC curre	nts using PSI			×				
Simulation name:	Simulation name: PSI AC Sweep							
Excitations								
<ul> <li>Use frequency</li> <li>Use sources de</li> </ul>			n project					
	Brows							
✓ Interpolate	e spectrum at mis	sing frequency	points					
Frequency Range	Setup							
Start Freq	g Stop Freq	Num. Points	Distribution					
1 10MHz	2GHz	100 Linear						
Add Above	Add Below	dd Below Delete Selection		review				
Save	Load	Set Default		ar Default				

- 2. In the Excitations area, ensure that Use sources defined in project is selected.
- 3. Set the Start Freq to 500 MHz.
- 4. Set the **Stop Freq** to **1 GHz**.
- 5. Set the **Num. Points/Step Size** to **50**. Please note that for each frequency point, a number of data files are written to the disk. The available disk space must be considered before specifying too many frequency points for simulation.
- 6. Set the **Distribution** to **Linear**.

Your settings should look like the following:

PSI AC Current Analysis 6-4

	Start Freq	Stop Freq	Num. Points	Distribution	
1	500MHz	1GHz	50	Linear	

7. Click Launch to begin the PSI AC current analysis.

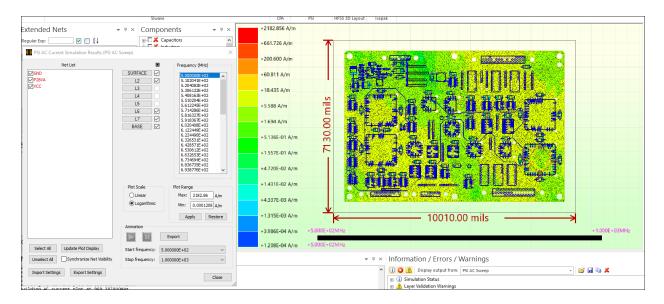
The **Messages** workspace updates to display a progress bar. When the analysis has completed, you can view AC currents as 2D plots.

### **Viewing AC Currents as 2D Plots**

To view the results plot:

1. Select the **Results** tab. In the **PSI** area, click **AC Currents** > **PSI AC Sweep** > **Plot Currents**.

The **Modeling** workspace updates to show the plot, and the **PSI AC Current Simulation Results** window appears:



PSI AC Current Analysis 6-5

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

2. In the Layer Toggle column ( $\square$ ), clear the check boxes for every layer except L2.

	×
SURFACE	
L2	
L3	
L4	
L5	
L6	
L7	
BASE	

- 3. In the **Frequency** list, click each frequency to view AC currents at that frequency.
- 4. Options in the **Animation** area allow you to view an animation of the plot at all frequencies, using the Play and Pause buttons.

### **Exporting Total Radiated Power**

After running the AC current analysis, the total power radiated from the PCB can be exported to a \*.csv file.

1. Click the **Results** tab. In the **PSI** section, click **AC Currents** > **PSI AC Sweep** > **Export** total radiated power.

2. Name and save the .csv file to any location.

PSI AC Current Analysis 6-7

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

3. View the saved file in any text editor.

PSI AC Current Analysis 6-8

1	"Freq [Hz]", "RadiatedPower [W]"
2	50000000,0.0564708489195
3	510204081.633,0.0316060131002
4	520408163.265,0.0226131402443
5	
6	
7	551020408.163,0.0406286155911
8	561224489.796,0.0626586399411
9	571428571.429,0.113268233374
10	
11	591836734.694,0.47488819016
12	
13	612244897.959,0.128254210618
14	622448979.592,0.0745450042714
15	632653061.224,0.0544804305976
16	
17	
18	663265306.122,0.0519442862736
19	· · · · · · · · · · · · · · · · · · ·
20	683673469.388,0.0837617742927
21	· · · · · · · · · · · · · · · · · · ·
22	704081632.653,0.208039800654
23	714285714.286,0.384086567773
24	724489795.918,0.506413955867
25	734693877.551,0.297220871799
26	744897959.184,0.164312436241
27	755102040.816,0.135141364912
28	765306122.449,0.105237767082
29	775510204.082,0.0892679461732
30	785714285.714,0.114190590754
31	795918367.347,0.201450244492
32	806122448.98,0.360715624804
33	816326530.612,0.357135076406
34	826530612.245,0.250732149771
35	836734693.878,0.175404953492
36	846938775.51,0.112408788204
37	857142857.143,0.0831272227951
38	
39	877551020.408,0.0894823899409
40	887755102.041,0.129584586183
41	897959183.673,0.213723150777
42	908163265.306,0.297288757457
43	918367346.939,0.243862625862
44	928571428.571,0.150671654475
45	938775510.204,0.0990916886915
46	948979591.837,0.0786709032629
47	959183673.469,0.0663596901453
48	969387755.102,0.054406081455
49	979591836.735,0.0496260801285
50	080705018 367 0 055217605505

PSI AC Current Analysis 6-9

4. In Slwave, click **FILE > Save** to save the project with the AC Current analysis result.

PSI AC Current Analysis 6-10

# 7 - Frequency Sweep of Voltages

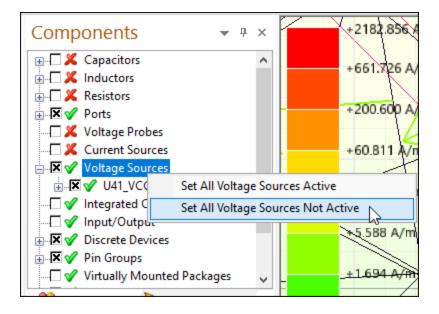
This section explains how to perform the following tasks:

- Disabling Voltage Sources
- Creating a Current Source on a component
- Placing a Voltage Probe in a region of interest
- Calculating a Frequency Sweep of Voltages
- Viewing a Voltage Probe plot

### **Disabling Voltage Sources**

The voltage source used in the AC Current analysis should be disabled, so that it does not interfere with the voltage frequency sweep.

- 1. In the Components workspace, ensure that Voltage Sources is selected.
- 2. Right-click on Voltage Sources and select Set All Voltage Sources Not Active.



The green check mark next to Voltage Sources becomes a red X.

U41\_VCC\_VSRC

### **Creating a Current Source on a Component**

The Frequency Sweep analysis requires one or more sources as excitations.

Frequency Sweep of Voltages 7-1

To define a Current Source:

1. Click Tools > Generate Circuit Element on Components.

The Circuit Element Generation window appears.

- 2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230\_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
- 3. In the Reference Terminal Component section, select Same as Positive Terminal.
- 4. In the Circuit Element Positive Terminal section, expand Pin Groups and select U41\_ VCC\_Group.
- 5. In the Circuit Element Reference Terminal section, expand Pin Groups and select U41\_ GND\_Group.
- 6. For the Circuit Element Type, select Current Source.

Your settings should look like the following:

Circuit Element Generation Dialog				-		×
Positive Terminal Component		Reference Terminal Compo	nent			
Part Name: MACH230 SMSOCKETAMD	~		MSOCKETAMD			$\sim$
Ref Des: U41		Ref Des: U41				~
UT UT		Same as Positive Termin	al			
		Joane as rosave remain				
Circuit Element Positive Terminal	Circuit Element Reference	Terminal	Circuit Elements			
Pin Groups Pin Groups P: U41 GND Group CLK_14 GND Group CLK_14 CLK_125K FORCEOFF GND HLC_AVAILABLE HLC_COMPLETE HLC_CTL HLC_CTL HLC_SOLATE HLC_ISOLATE HLC_RRC_RDY HLC_RRC_RDY HLC_START HLC_START HLC_STOP TCI1_LS TCI1_ON_RED TCI1_POWER_AVAILABLE TCI1_POWER_ISOLATE TCI2_DIG1_DN_GWITCH TCN_DIG1_DN_GWITCH TCN_DIG1_DN_GWITCH Expand Collapse Find Pin at Location	Pin Groups  Pin Groups  Pin Groups  Put1_GND_G  Put1_VCC_Gr  GND  FORCEOFF GND  HLC_AVAILABLE HLC_CTL HLC_CTL HLC_ERR HLC_SOLATE HLC_SOLATE HLC_RRC_RDY HLC_RRC_RESET HLC_RST_N HLC_ST_N HLC_ST_N HIC_SHIFT  Expand Collapse  Use nearest pin as ref Group pins within the Circuit Element Type Capacitor Inductor Database	Find Pin at Location Ference pin reference distance O Port O Current Source	Current Sources Inductors Resistors Voltage Probes Voltage Sources Terminals			
Reg Expression syntax {Net name}:{Pin name} Reg exp:	O Resistor O S-Param Cir Elem O Terminal	<ul> <li>Voltage Source</li> <li>Voltage Probe</li> </ul>	Delete		Edit	
	Naming Convention	Create		OK	Car	ncel

7. Click Create.

The Set Current Source Properties window appears.

Frequency Sweep of Voltages 7-3

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

Set (	Current Source Properties			×			
Nam	ne: U41_VCC_ISRC						
۹	Frequency Independent Parameters						
	Magnitude: Parasitic Resistance:	1E-5 5E+07	Amps Ohms				
	Phase:	0	Degrees				
O Frequency Dependent							
	Path: Browse						
	File should contain data in <freq> <real> <imag></imag></real></freq>						
		0	OK	Cancel			

- 8. Shorten the name to U41\_VCC\_ISRC.
- 9. Set the Frequency Independent Magnitude to 1E-5 Amps.
- 10. Ensure that the Parasitic Resistance is set to 5E+07 Ohms.
- 11. Ensure that the **Phase** is set to **0** Degrees.
- 12. Click **OK** to accept the current source definition.
- 13. Click **OK** to close the **Circuit Element Generation** window.

### **Creating a Voltage Probe**

The problem of induced ripple voltage in the Power Distribution Network (PDN) can be investigated by measuring the voltage between the VCC and GND nets, using a voltage probe placed at a location of interest on the PCB.

You will place a probe at the lower-right corner of the PCB, where U41 is located. The X- and Y- coordinates for the probe are X=8500mil, Y=0mil.

Perform the following steps:

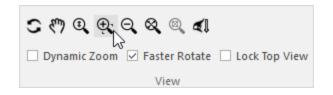
1. Make sure the **Units** field in the menu bar is set to mils.

x:	8500	y:	0	z:		dx:		dy:		Units:	mils 🔻	
----	------	----	---	----	--	-----	--	-----	--	--------	--------	--

Frequency Sweep of Voltages 7-4

Ansys Electromagnetics Suite 2022 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential

2. Select the **View** tab. Use the **Zoom In** icon (or your mouse wheel) to enlarge the lower-right corner of the PCB.



- 3. Select the **Home** tab.
- 4. In the Circuit Elements area, select the Add Voltage Probe icon (<sup>1</sup>U<sup>1</sup>).

The cursor changes to a small bullseye.

- 5. Use the location indicators at the bottom-right of the screen to position your cursor at x: 8500 mils, y: 0 mils.
- 6. Click the left mouse button twice, slowly each time. The first click locates the positive terminal, the second click is for the reference terminal. The terminals are located in the same position since these probes measure the voltage between two planes.

			_	-	
Layer	Net	^	Layer	Net	^
SURFACE			SURFACE		_
L2	GND		L2	GND	
🔲 L3			L3		
L4			L4		
L5			L5		
L6		_	L6		
L7	VCC	~	L7	VCC	~
<		>	<		>

The Select layers for voltage probe terminals window appears.

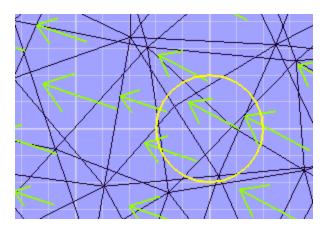
- 7. For the Positive Terminal Resides on Layer, select L7 (VCC).
- 8. For the Negative Terminal Resides on Layer, select L2 (GND).
- Click OK to close the Select layers for voltage probe terminals window.
   The Edit Probe Name window appears.

Frequency Sweep of Voltages 7-5

Edit pro	be name			×
Name:	VPROBE1			
		ОК	Cancel	

10. Enter **VPROBE1**, and click **OK**.

The voltage probe (a yellow circle in the following figure) appears.



### **Running a Frequency Sweep**

Frequency sweep calculates the voltages across the VCC and GND planes at the locations of the probe. The Slwave frequency sweep shows how the current drawn by a device can lead to voltage ripple in the power distribution system.

Run the frequency sweep analysis:

1. Click Simulation > Compute Frequency Sweeps...

The Compute Frequency Sweep window appears.

- 2. For excitations, select Use sources defined in project.
- 3. Set the Start Freq to 500MHz.
- 4. Set the **Stop Freq** to **1GHz**.
- 5. Set the Num. Points to 200.
- 6. Set the **Distribution** to **Linear**.
- 7. Set the Voltage Surface Plot Options to plot voltage difference between layers L2 and L7.

Your settings should look like the following:

Frequency Sweep of Voltages 7-6

Compute	Frequency	Sweep			×			
Simulation	name:	Frequency S	Sweep 1		~			
Excitatio	ons							
	sources defir sources defir							
			Browse					
	✓ Interpolate spectrum at missing frequency points							
Frequer	ncy Range Se	tup						
	Start Freq	Stop Freq	Num. Points	Distribution				
1 50	00MHz	1GHz	200	Linear				
Add	Add Above A		Delete Selec	tion	Preview			
S	Save		Set Default 0		ear Default			
Voltage Surface Plot Options Plot voltage difference between planes Layer L2  v and Layer L7  v [reference layer (ground)								
Other sol	ver options	Save Se	ettings	Launch	Close			

8. Click Launch to begin the simulation.

The **Messages** workspace displays a progress bar that lets you know when the simulation is complete.

### **Plotting Probe Voltage**

To plot the voltages at the probe locations:

Frequency Sweep of Voltages 7-7

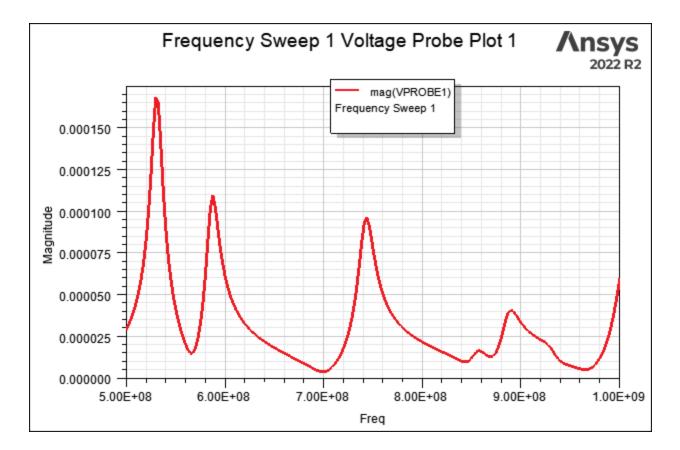
- 1. Select the **Results** tab.
- Click Frequency Sweep > Frequency Sweep 1 Results > Plot Probe Voltages. The Voltage Probe Plot Generation window appears.

Voltage Probe Plot Generat	ion (Frequency Sweep 1)	×
Plot name: Voltage Probe Plot 1		
Plot Options	Voltage Probes to Plot	
Plot magnitude	Plot Probe	
○ Plot phase	VPROBE1	
	Regular expression syntax: {PORT1}:{PORT2} Select	
	Regular expression: Unselect	í.
	Create Plot Close	
		_

3. Ensure that **Plot magnitude** and **VPROBE1** are selected.

#### 4. Click Create Plot.

Ansys Electronics Desktop launches. The **Reporter** window opens, displaying the plot of magnitude versus frequency.



The peaks in the voltage correspond to resonant mode 4, mode 8, and mode 10.