Getting Started with SIwave: 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.

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, http://www.ansys.com/Support. 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 2-3

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 **3-1**

Running the Resonant Modes Analysis 3-1

Viewing Resonant Modes Analysis Results 3-2

4 - SIwave SYZ Analysis **4-1**

Defining Pin Groups for GND and VCC 4-1

Defining a Port Between Pin Groups 4-3

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

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

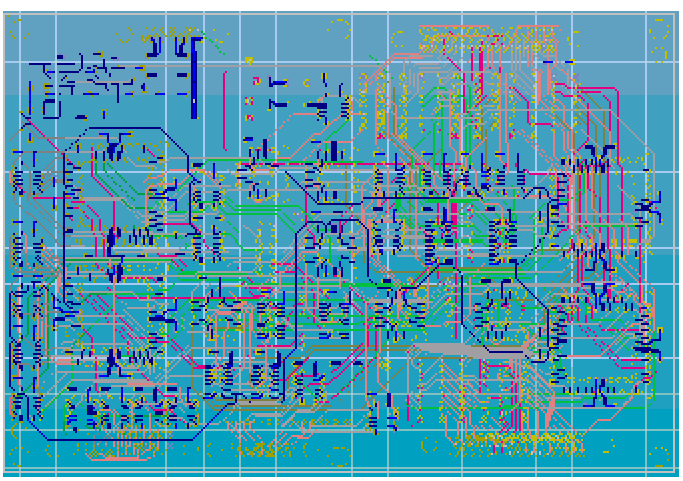
This Getting Started Guide is intended to quickly familiarize you with the capabilities of SIwave. This guide leads you step-by-step through importing a PCB design, setting up and performing three SIwave analyses, and viewing the results of the simulations.

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

- Importing a geometric PCB model
- Validating the design
- Running three SIwave 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.



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

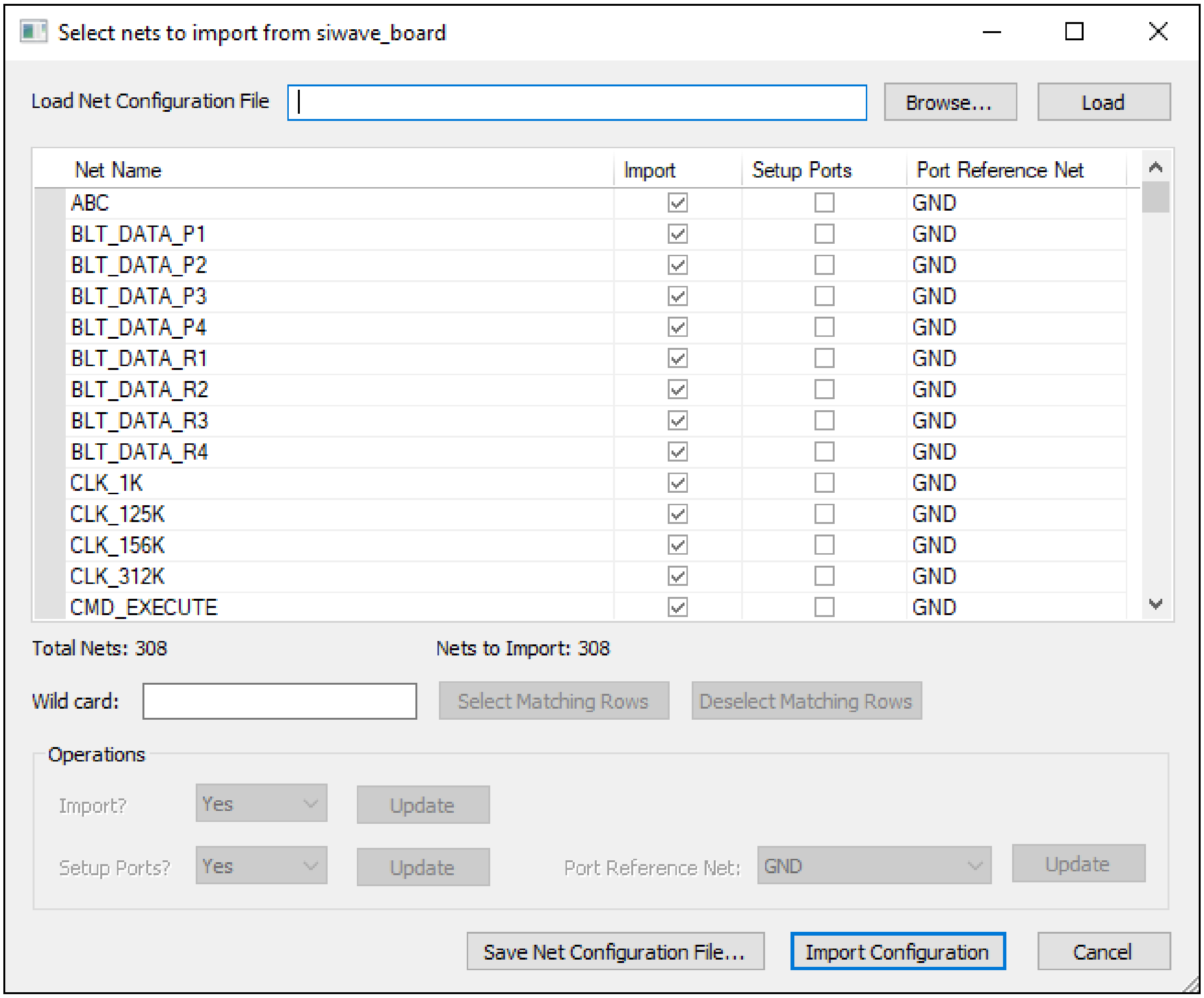
Importing and Saving the Project

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 **SIwave**.
2. If the **Welcome to SIwave** 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\SIwave**
   - Linux: **/Program Files/AnsysEM/v222/Linx64/Examples/SIwave**
6. Select the file **siwave_board.anf**.
7. Click **Open**.

   The **Select nets to import from siwave_board** window appears.

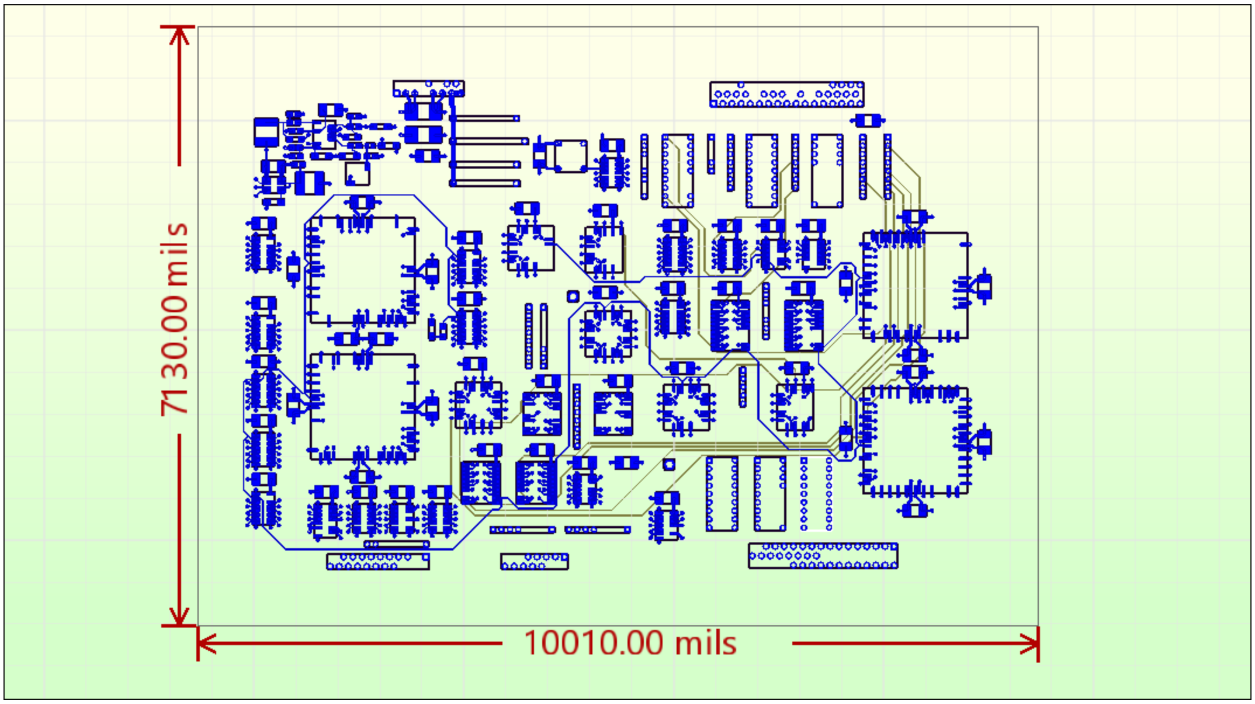


8. Leave the settings as-is, and click **Import Configuration**.

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

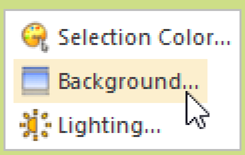
   If the **SIwave Workflow Wizard** opens, close it to reveal the Modeling workspace.

   The design should look like this:



Note:

You can change SIwave's background colors from the **View** tab:



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**.

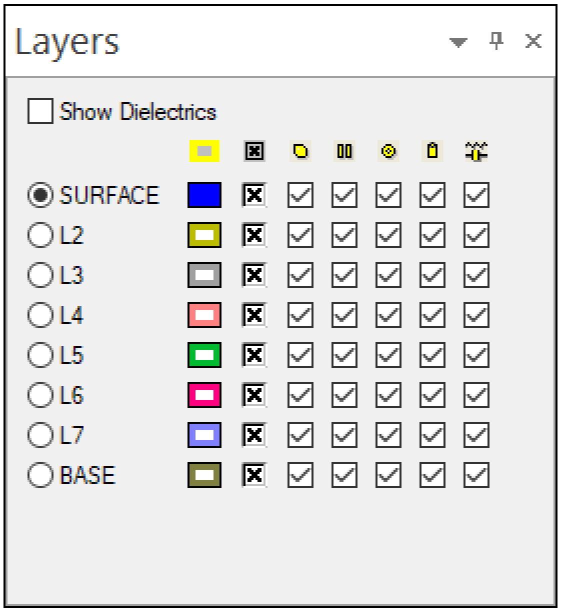
Setting PCB Element Visibility

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

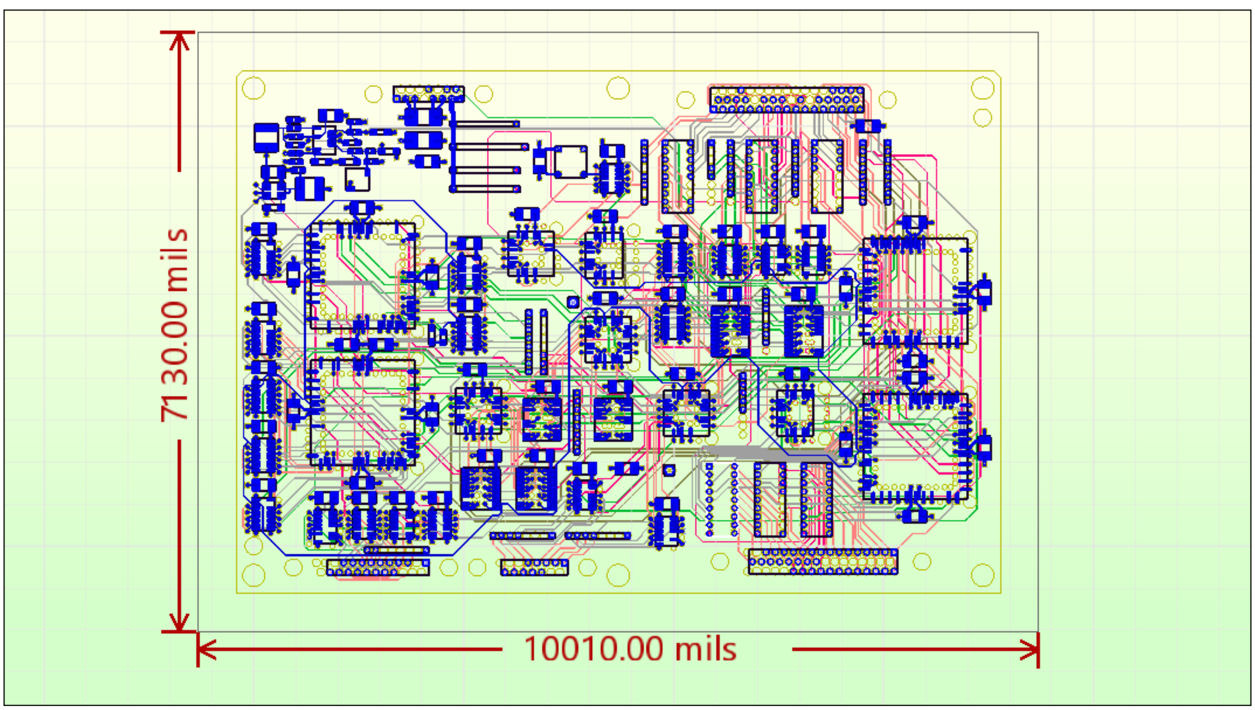
To turn on visibility for geometric elements (planes, traces, pads, vias, and circuit elements):

1. Navigate to the **Layers** workspace.
2. 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:



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



Note:

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

Viewing the Layers Workspace and Layer Stackup

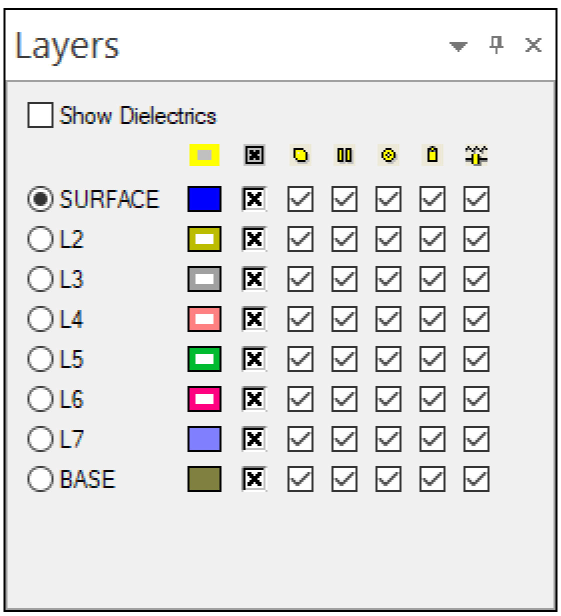
The **Layers** workspace and the **Layer Stackup Editor** control the visibility and properties of the package layers.

Layers can be viewed in either outline or filled mode.

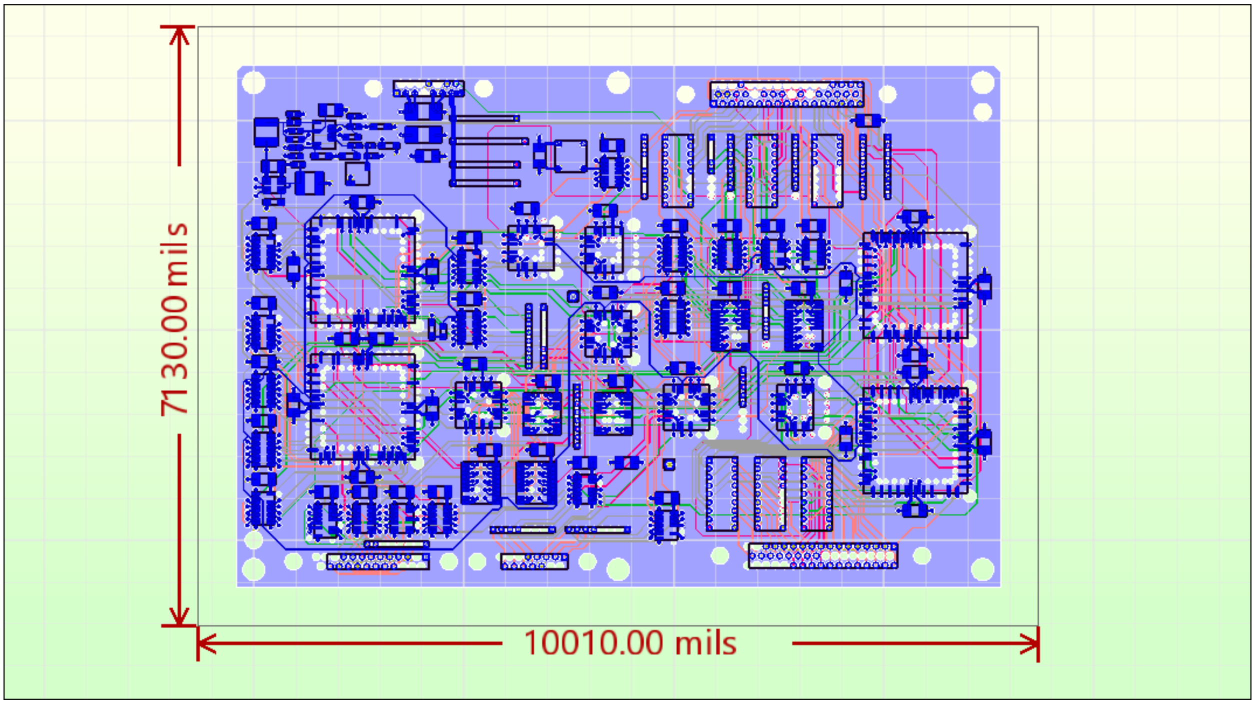
To set the layers for this project to filled mode:

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:



The Modeling workspace should look like the following:

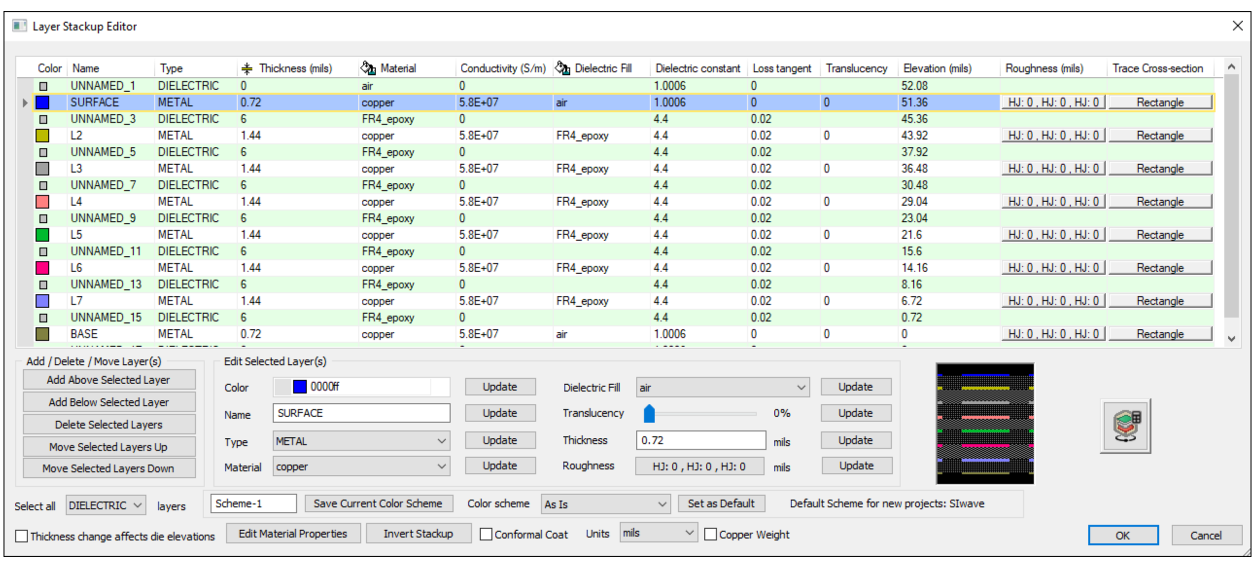


Viewing the Layer Stackup

To view the layer stack:

1. Navigate to the **Home** tab.
2. In the **Edit Tools** area, click **Layer Stackup Editor**.

   The **Layer Stackup Editor** appears.



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.

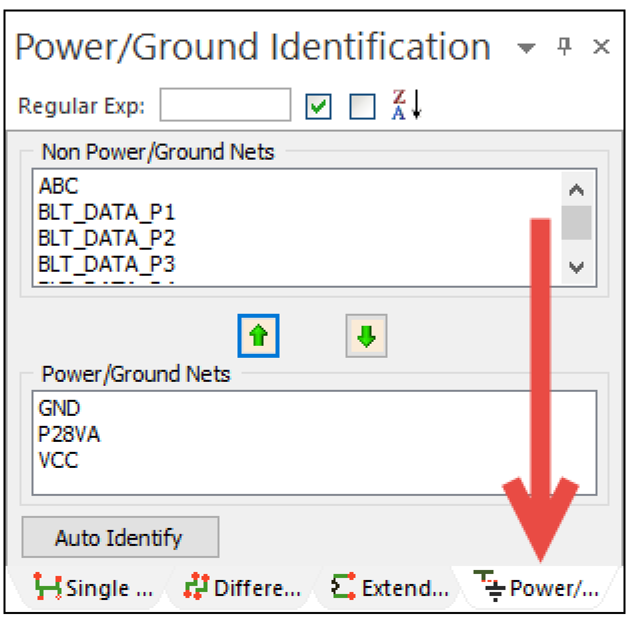
> **Note:**
>
> By default, the **Nets** workspace is located on the upper-left side of the SIwave window. Workspaces can be moved by dragging and dropping them to another location.

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

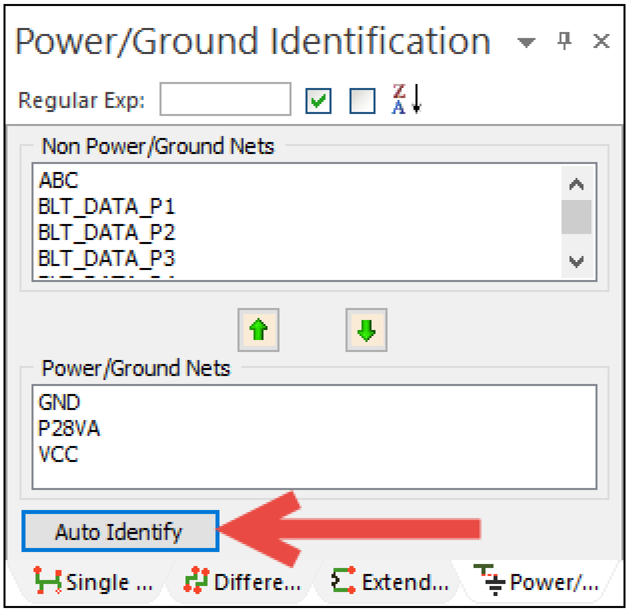
Nets containing large planes must be classified as Power/Ground nets. Signal nets containing microstrip and stripline routing need to be classified as Non Power/Ground nets. This enables the solver to judiciously choose the mesh refinement and optimization strategies for the signal and power/ground nets.

To identify power/ground nets:

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



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



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

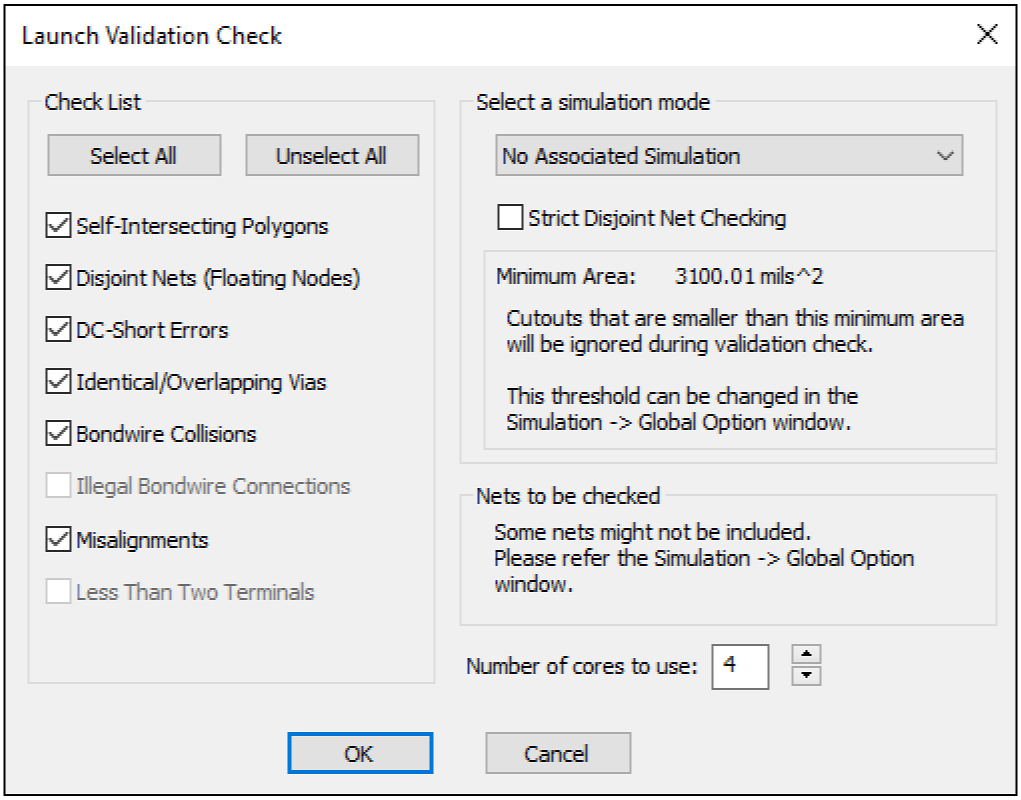
3. If any net is incorrectly identified, click to highlight the net name and use the up and down arrows to move it to the correct list.

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.

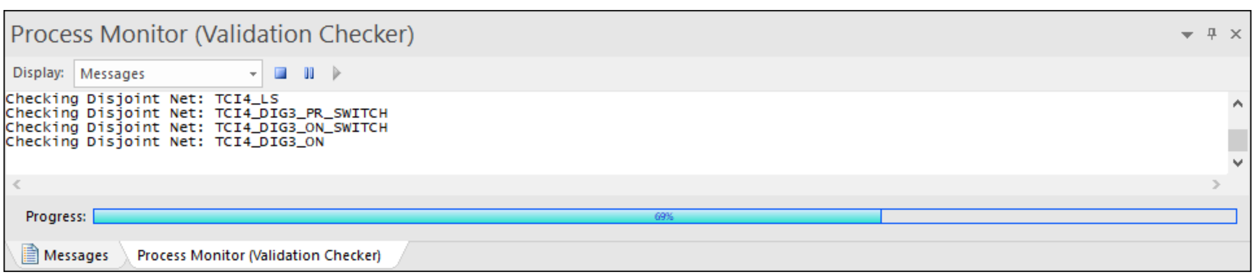
1. Click **Tools**. In the **Inspection Tools** area, click **Validation Check**.

   The **Launch Validation Check** window appears.

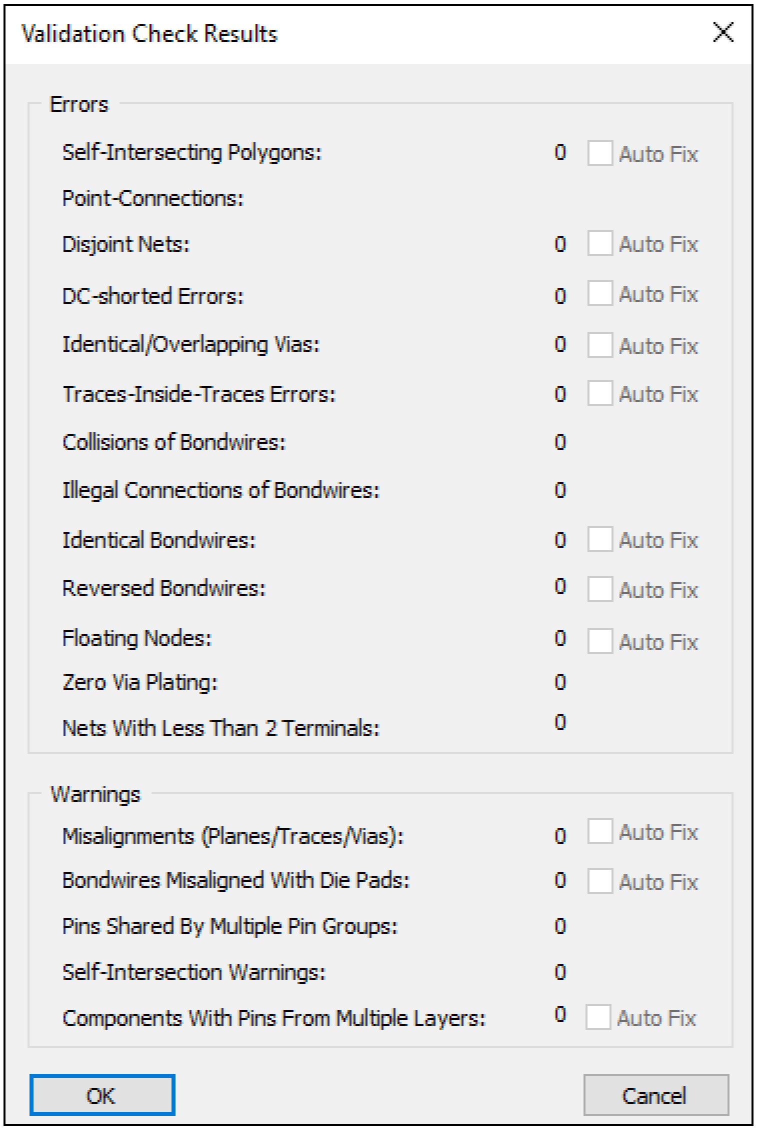


2. Leave the default settings, and click **OK** to start the check.

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



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



If there were errors or warnings that SIwave 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.

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

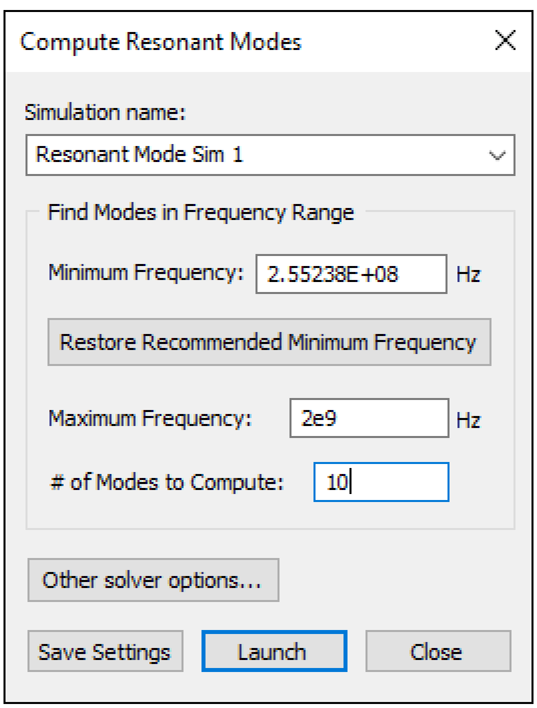
Running the Resonant Modes Analysis

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 SIwave Resonant Modes analysis:

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

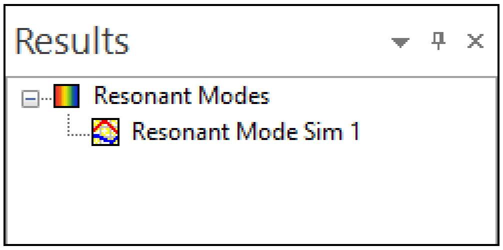
   The **Compute Resonant Modes** window appears.
2. Verify that **Minimum Frequency** is set to 2.55238E+08. If it is not, click **Restore Recommended Minimum Frequency**.
3. Set the **Maximum Frequency** to **2e9** (2 GHz).
4. Set the **# of Modes to** compute to **10**.



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:



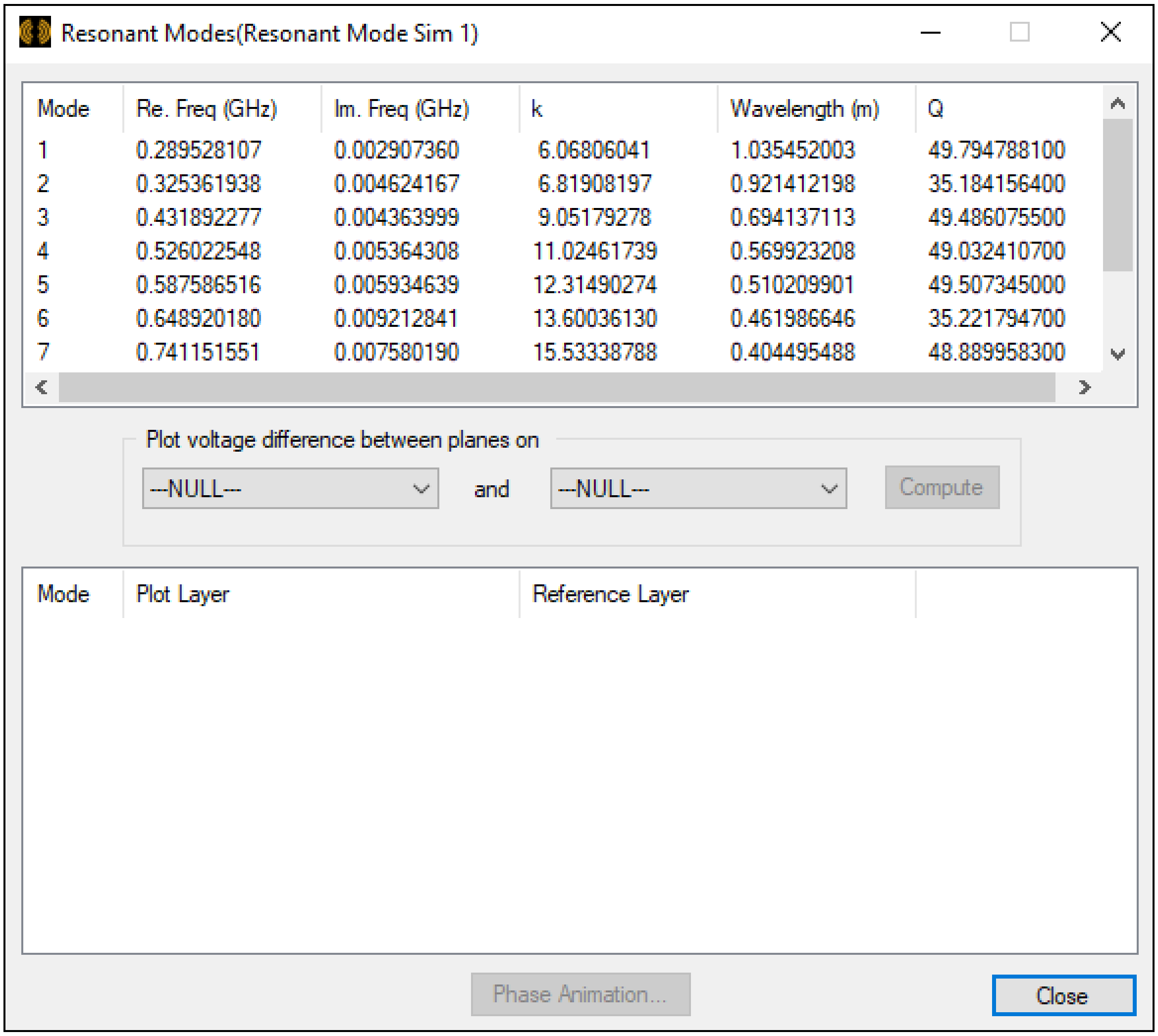
Viewing Resonant Modes Analysis Results

After the analysis has finished, Resonant Modes results can be viewed in a table, or as two-dimensional 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.

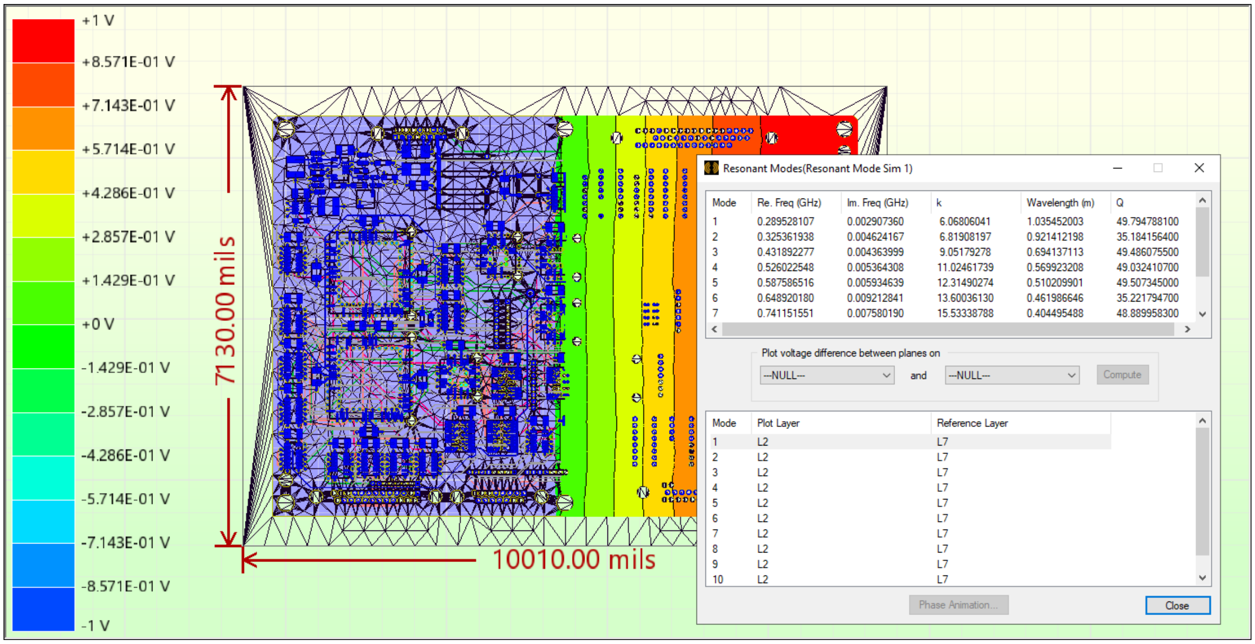


At the center of this window, two drop-down menus allow you to compute the voltage differences between two planes.

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

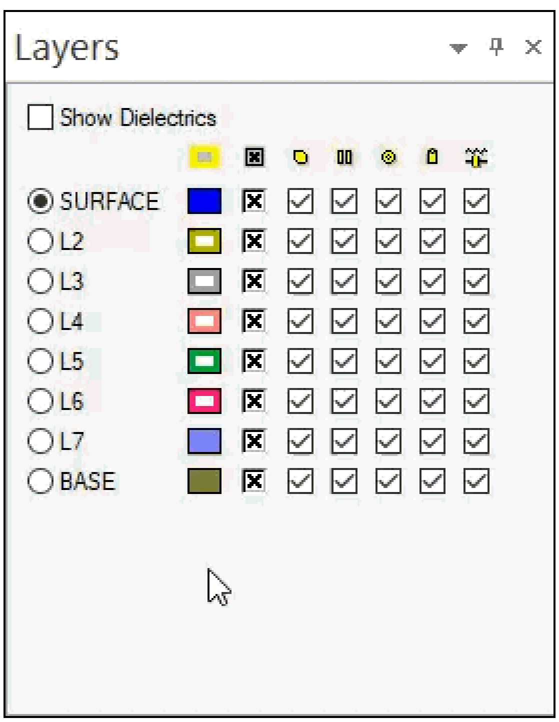
1. Use the drop-down menus under **Plot voltage difference between planes on** to select **L2** and **L7** respectively.
2. Click **Compute** to generate the 2D plot data from the solution data.

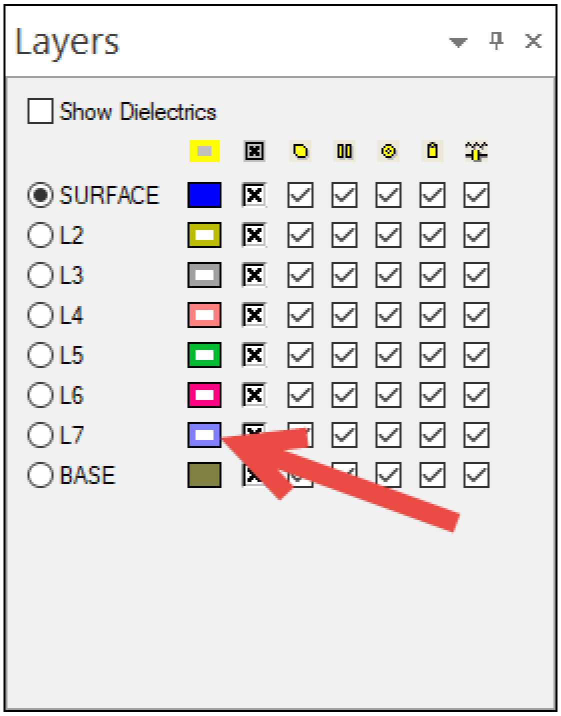
   The lower part of the window updates to display the modes available for plotting. There are ten modes, and the underlying Modeling workspace updates to display Mode 1.



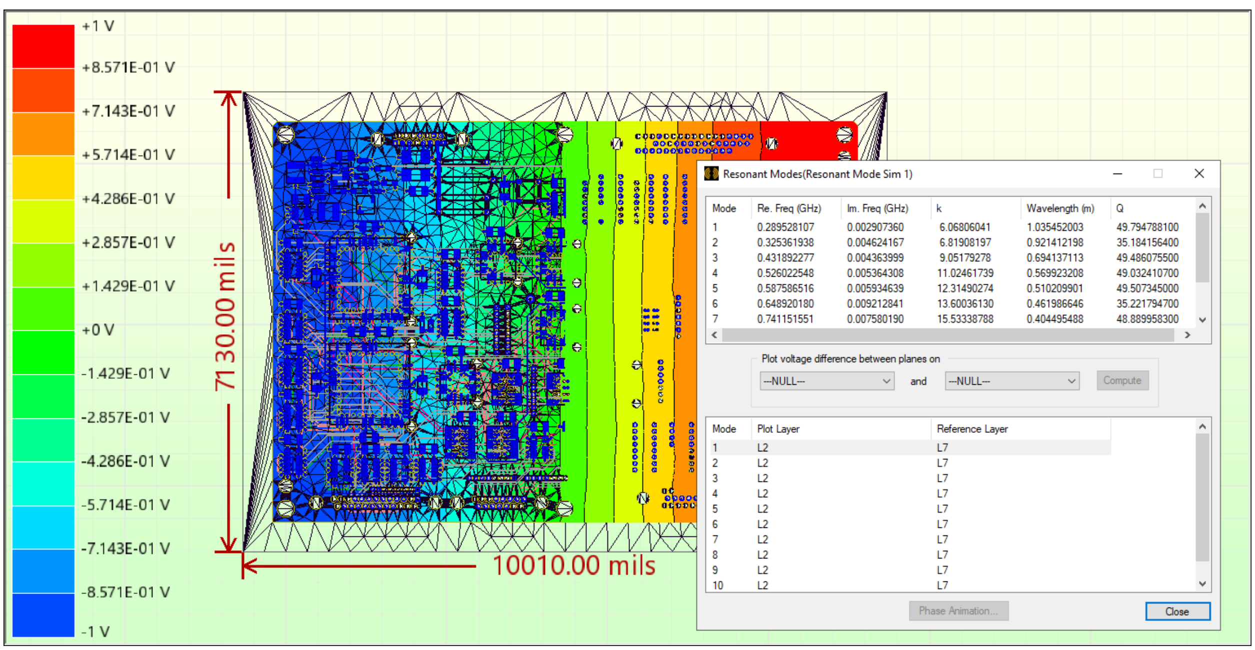
You'll notice that layer L7's display mode obscures part of the plot.

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





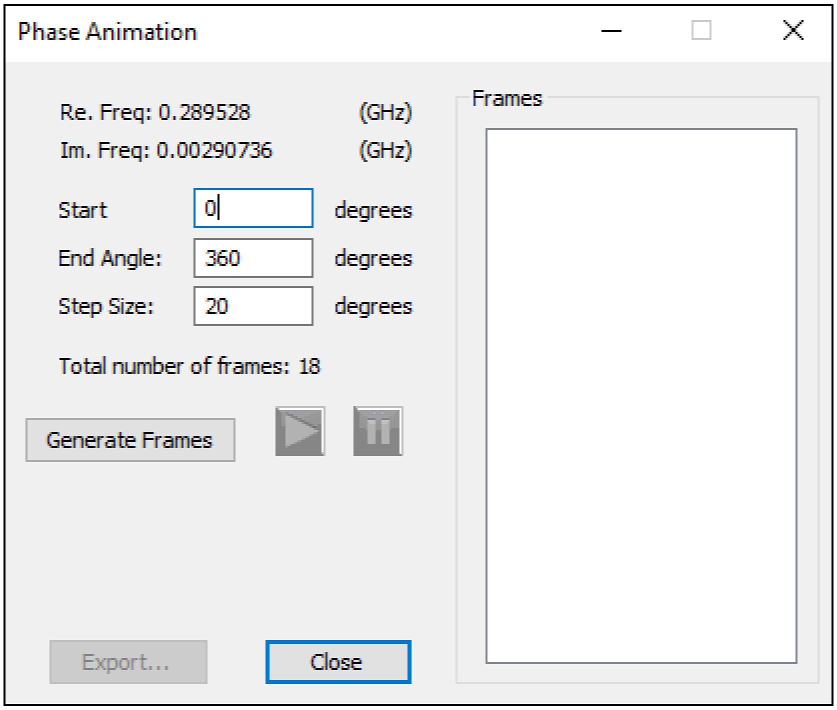
The entire plot is now visible:



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.

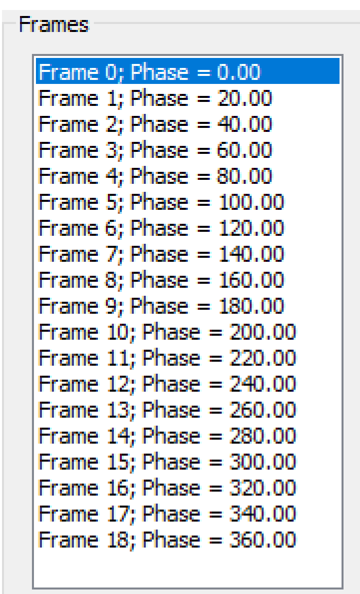
5. To view the modes in sequential animation, click **Phase Animation**.

   The **Phase Animation** window appears.



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

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



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.

4 - SIwave SYZ Analysis

This section explains how to perform the following tasks:

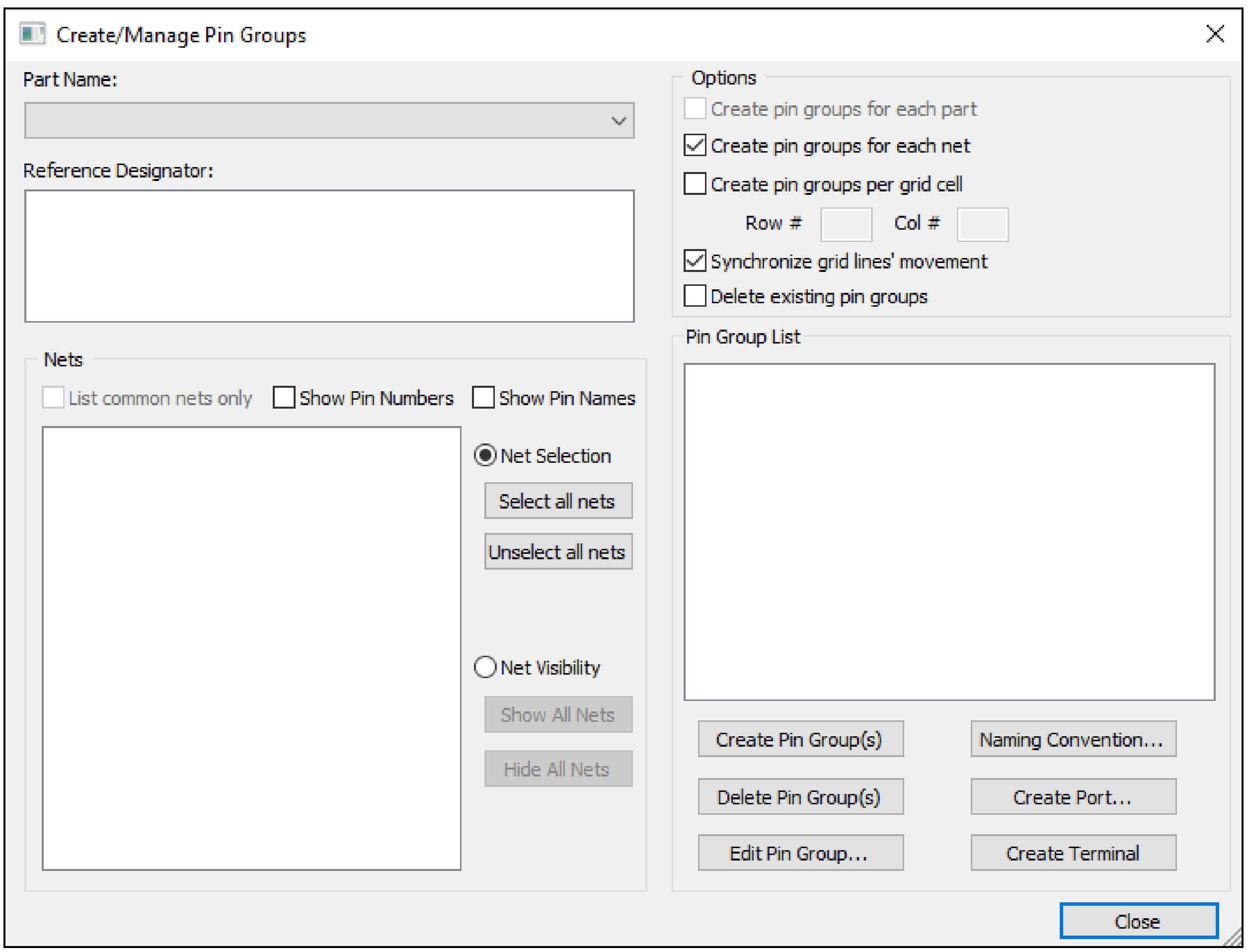
- Defining pin groups
- Defining a port between the pin groups
- Generating the SYZ-parameters for selected power supply nets
- Viewing the frequency-dependent impedance of the planes on an X-Y plot

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.



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

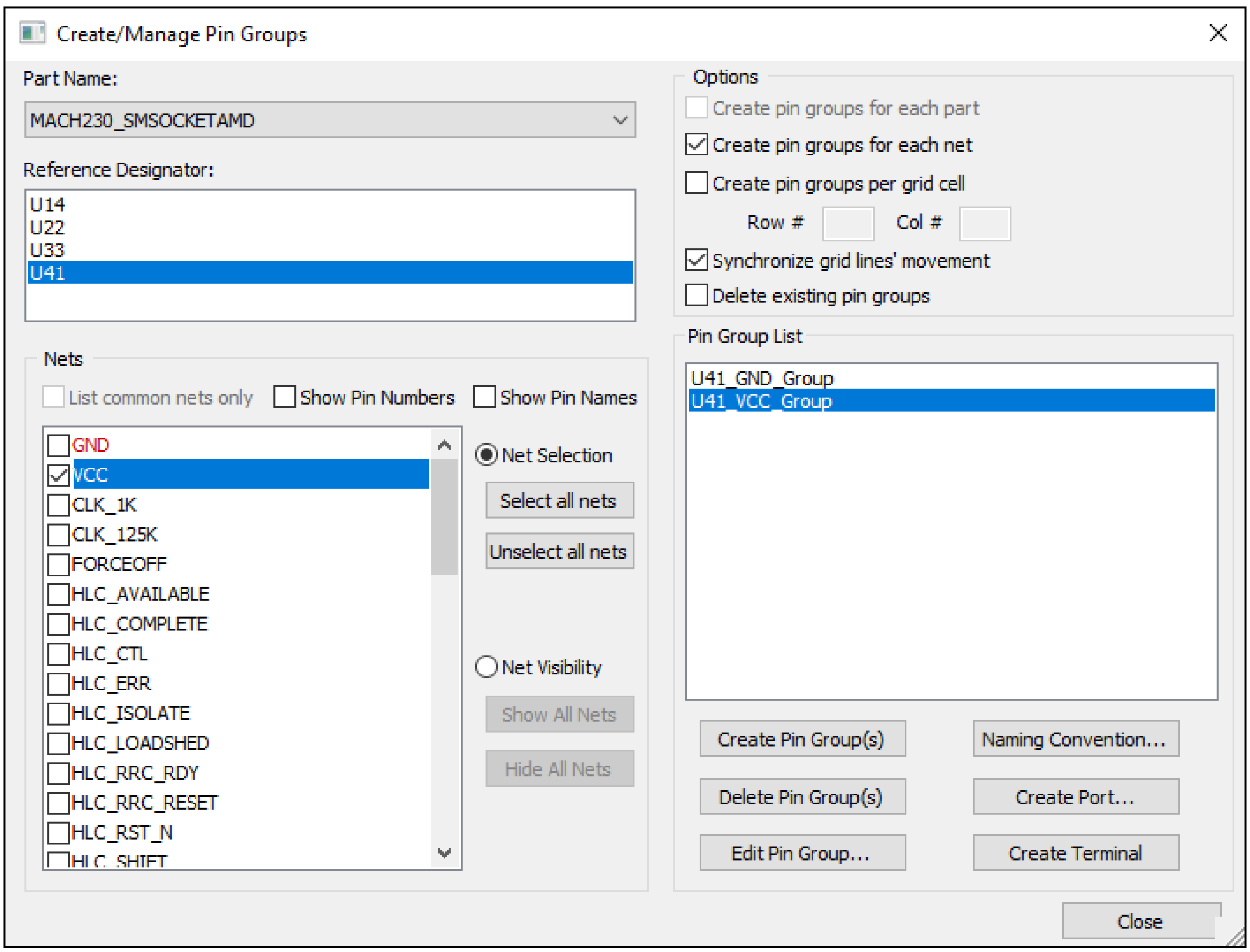
1. From the **Part Name** menu, select **MACH230_SMSOCKETAMD**.
2. From the **Reference Designator** list, select **U41**.
3. In the **Nets** list, ensure that the check box next to **GND** is selected, and that no other nets are selected.
4. Click **Create Pin Group(s)**.

   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**.



3. Click **Close**.

Defining a Port Between Pin Groups

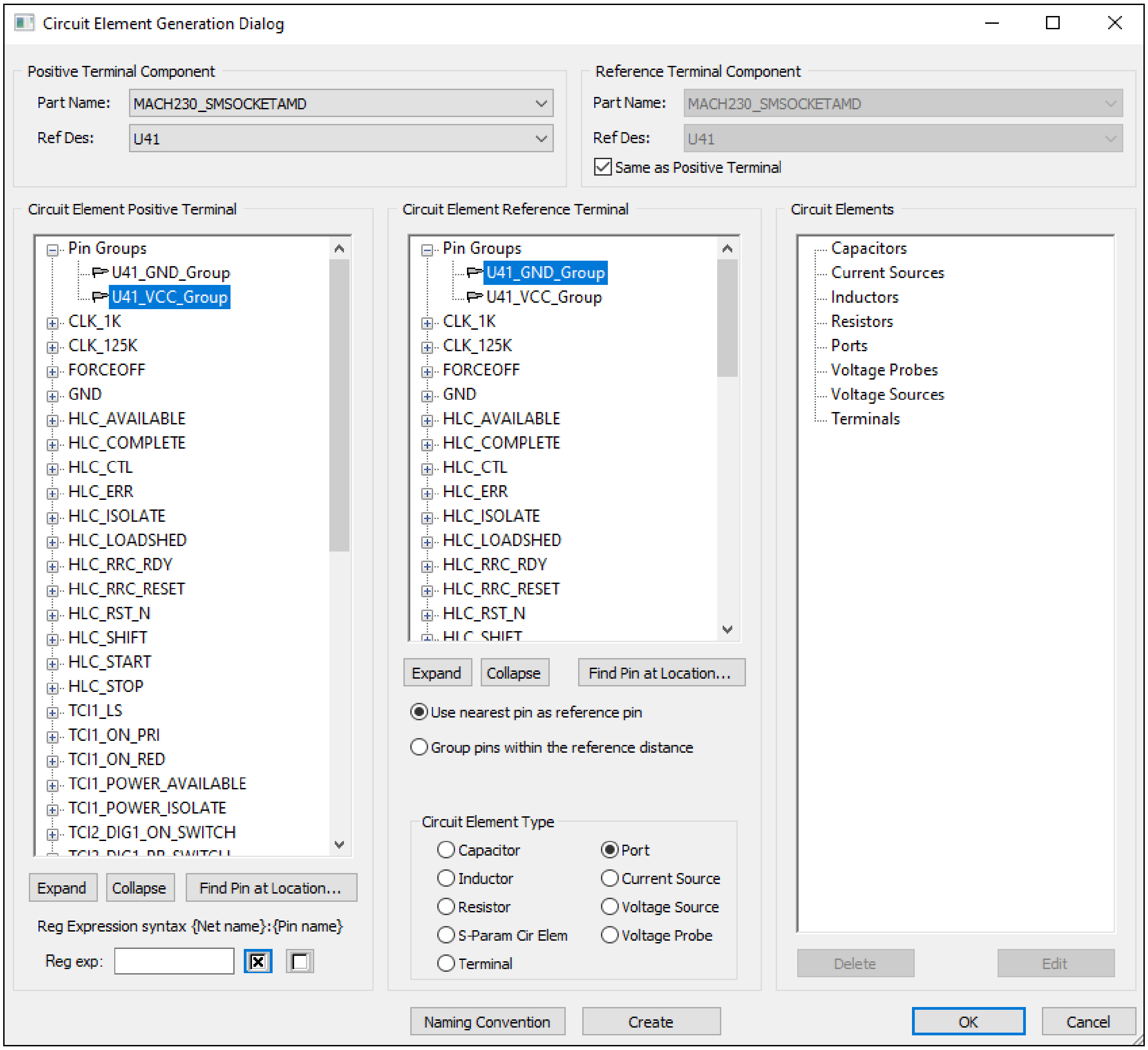
Define a port on component U41.

1. Click **Tools** > **Generate Circuit Element on Components**.

   The **Circuit Element Generation Dialog** appears.

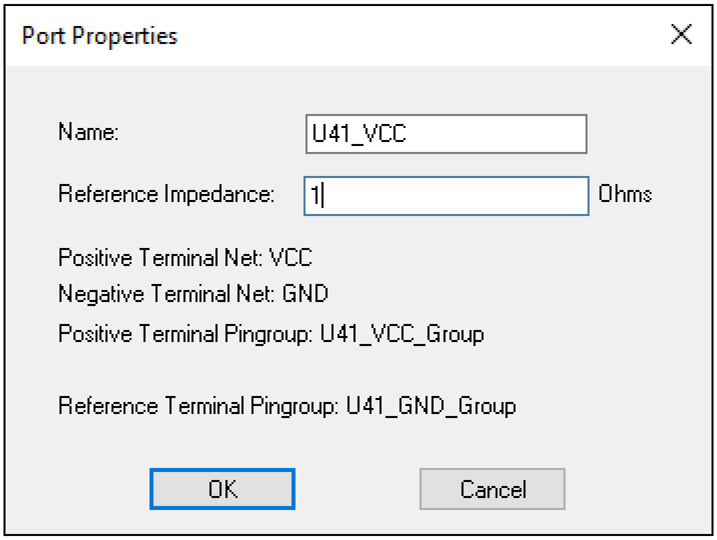
2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
3. In the **Reference Terminal Component** section, enable the **Same as Positive Terminal** check box, which will populate the 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.
6. In the **Circuit Element Type** section, select the **Port** radio button.

   Your selections should look like the following:



7. Click **Create**.

   The **Port Properties** window appears.



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

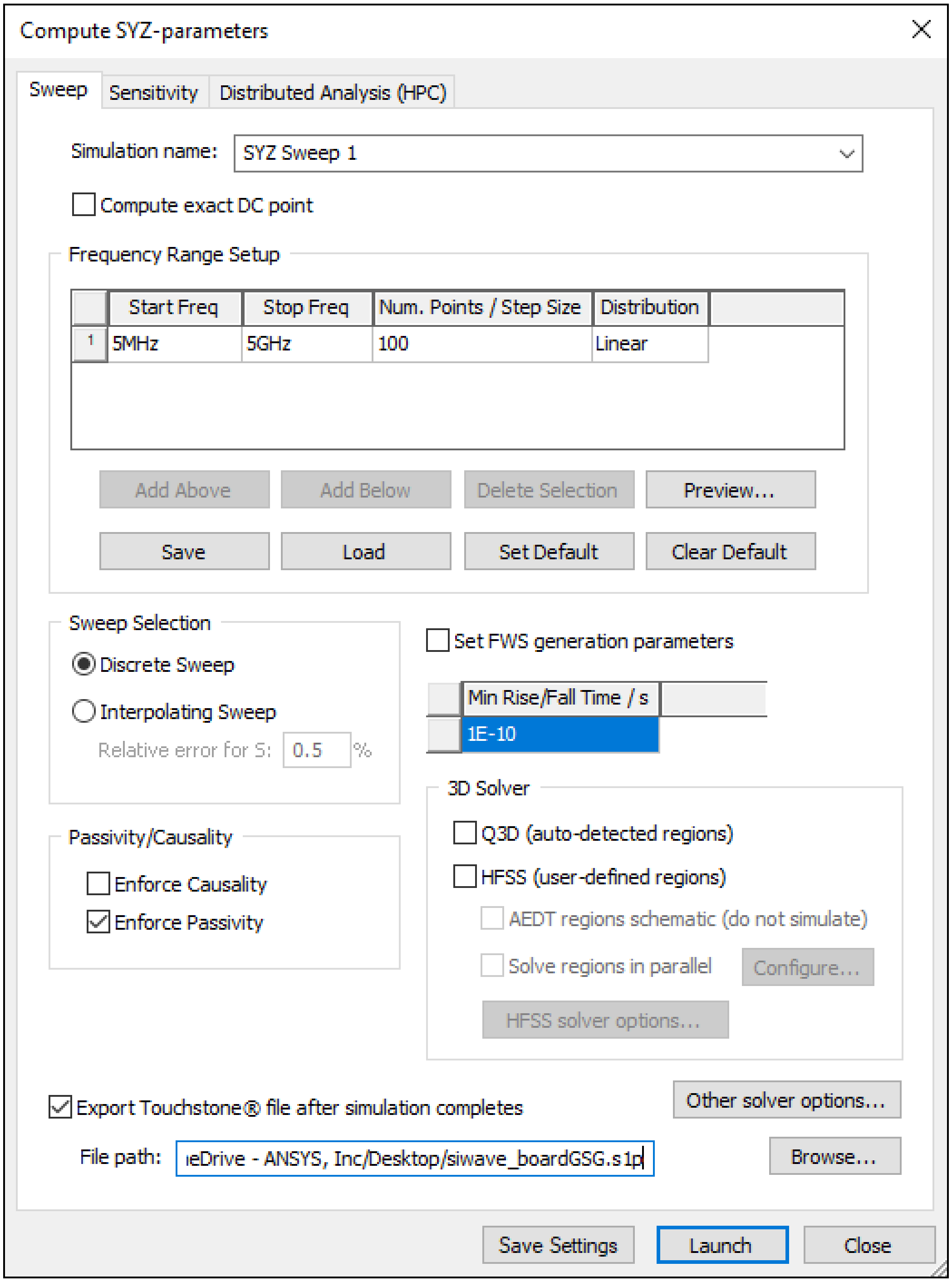
Next, we calculate the frequency-dependent impedance response at port U41_VCC.

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 **SIwave** 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.

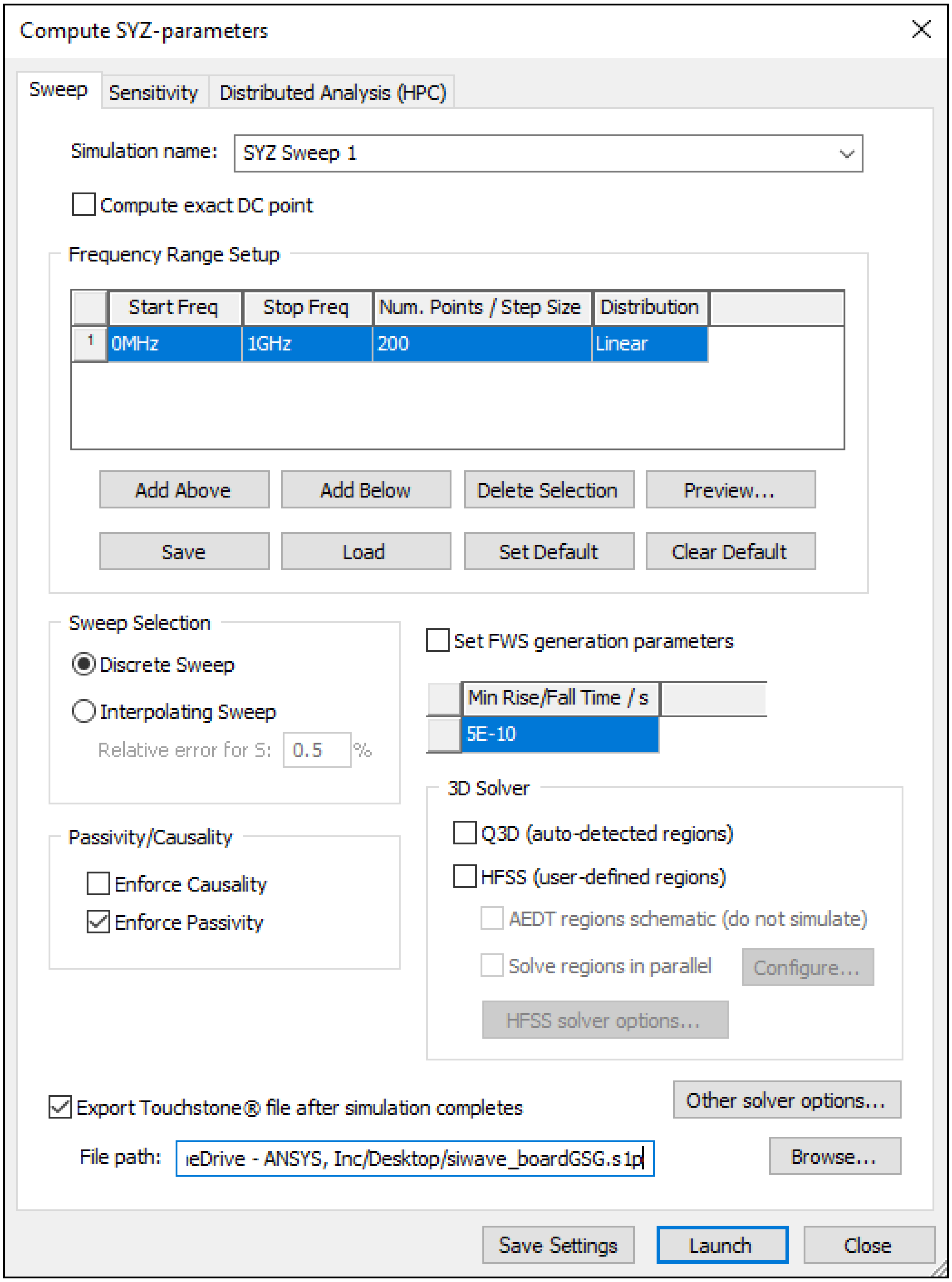
The **Compute SYZ-parameters** window appears, populated with default settings.



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**.

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:



8. Click **Launch** to launch the SYZ analysis.

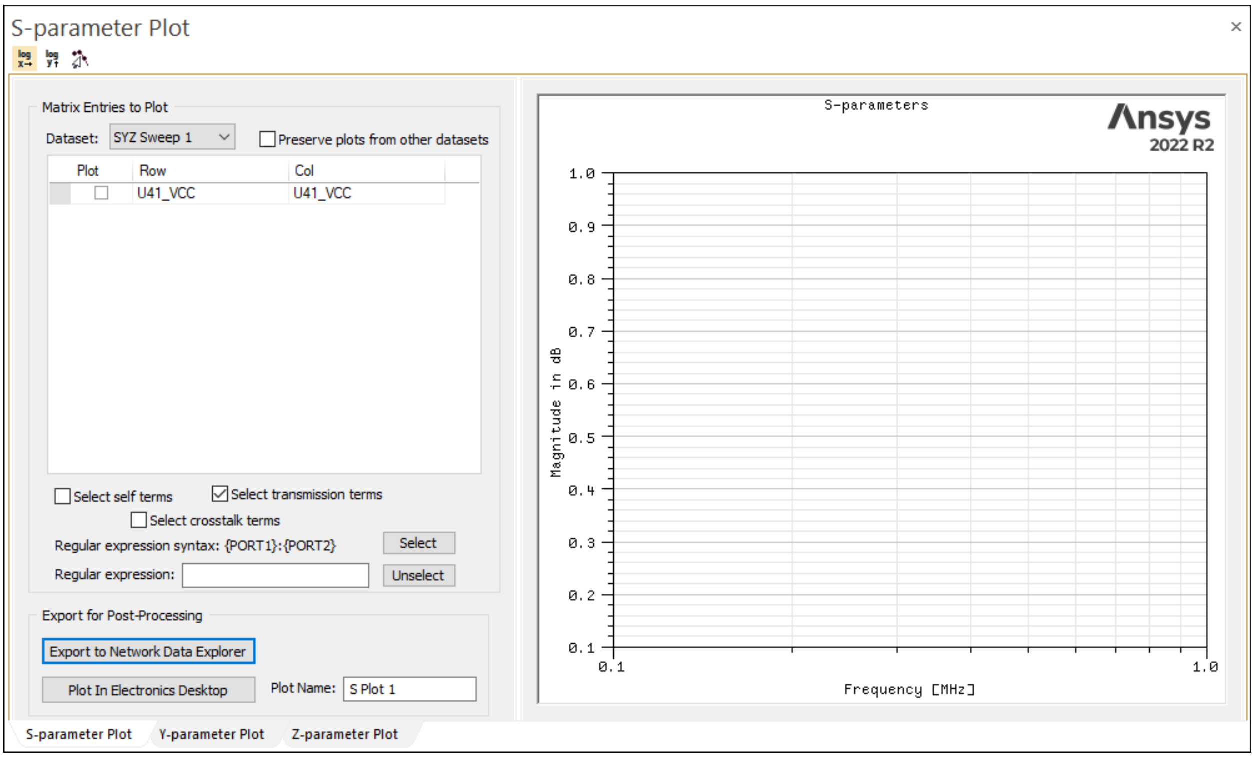
The **Messages** workspace updates with a progress bar showing simulation progress. When it has finished, you can view the impedance response.

Viewing Impedance Response

To view the frequency-dependent impedance response:

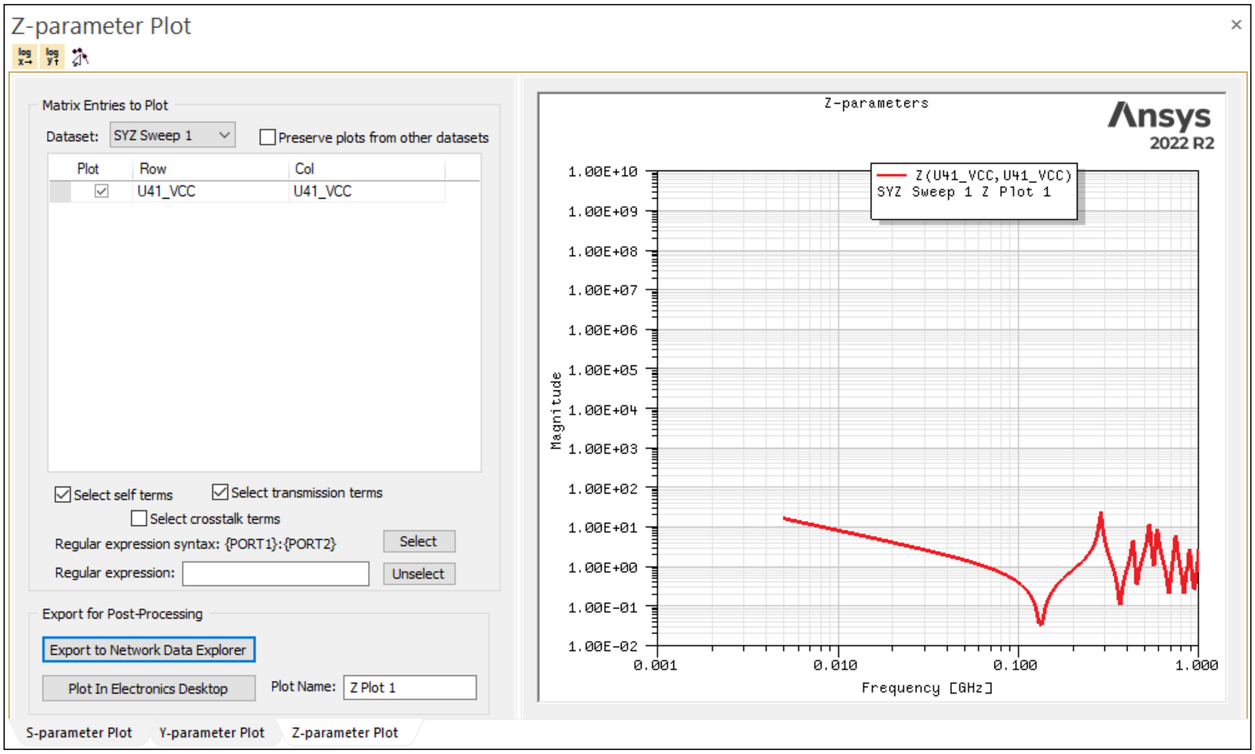
1. Select the **Results** tab. In the **SIwave** area, click the **SYZ** icon to access its drop-down menu. Select **SYZ Sweep 1** > **Plot Magnitude...**.

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



2. Click the **Z-parameter Plot** tab at the bottom of the window.

   The window changes to **Z-parameter Plot**.



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.

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 (SIwave SYZ Analysis), please do so to set up your pin groups and ports.

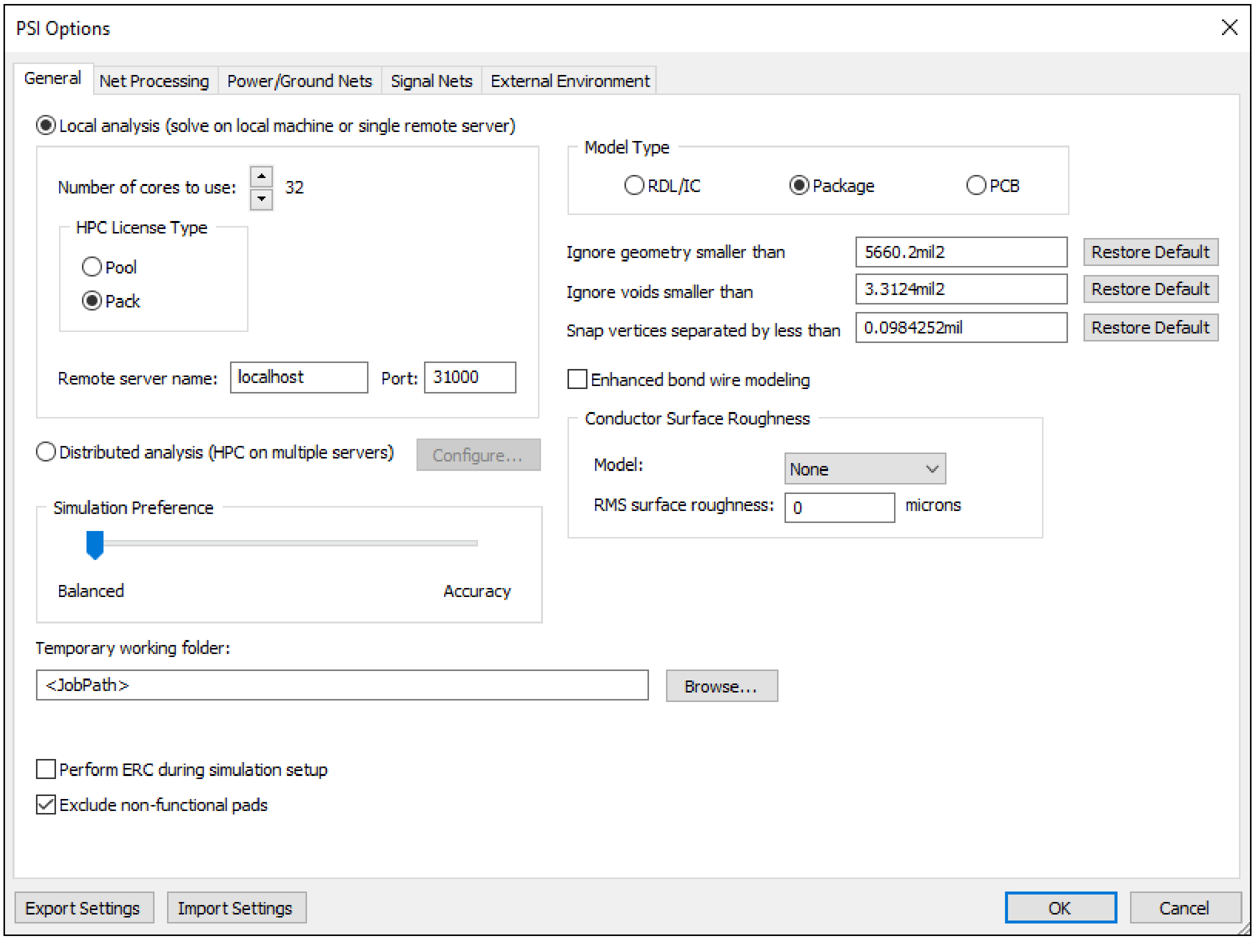
Generating SYZ Parameters using PSI

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.

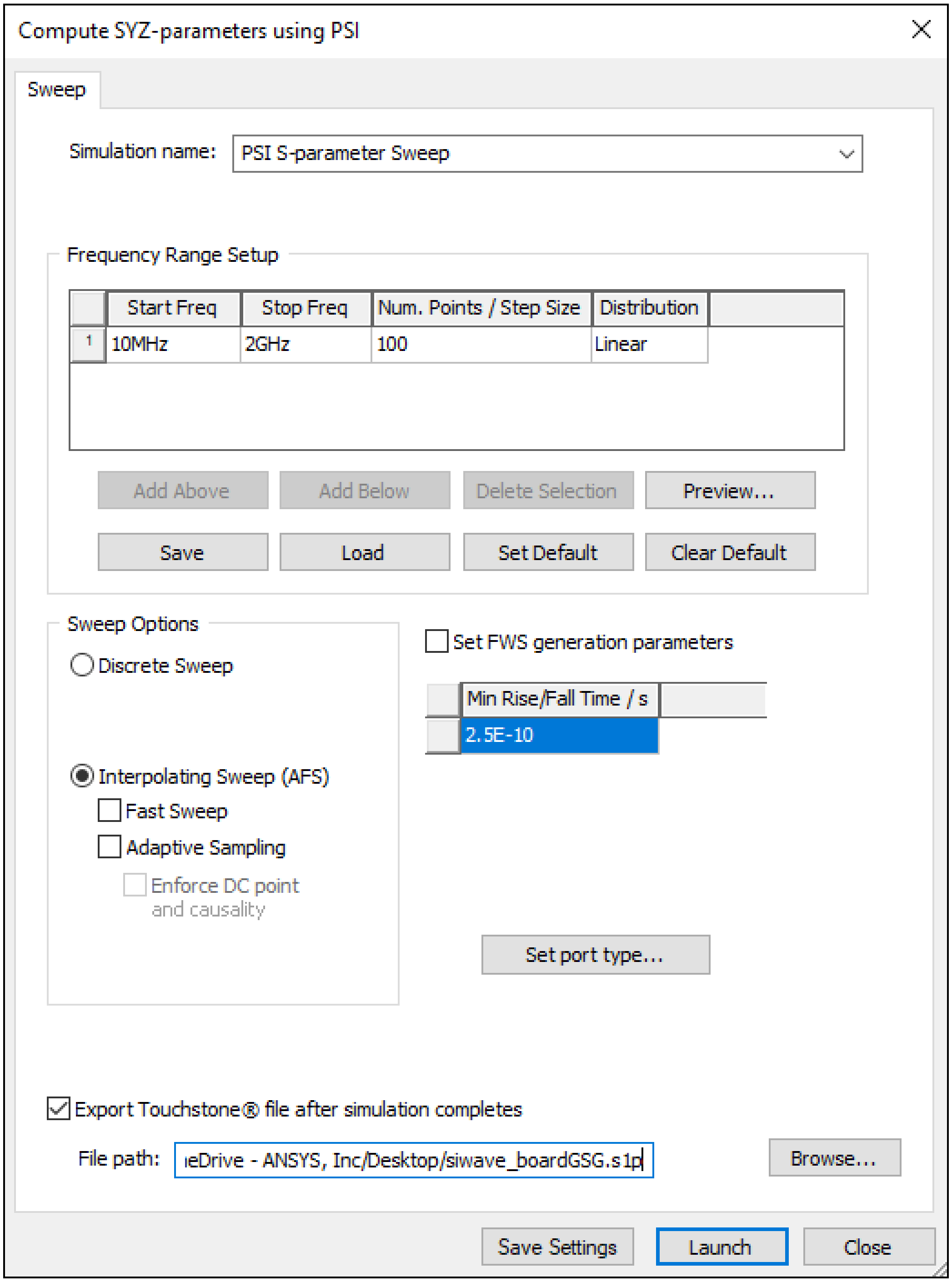


2. Ensure that **Local Analysis** is selected. In the box below, use the up and down arrows to change the **number of cores to use**. To use all cores, click the up arrow until you reach the maximum.

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

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

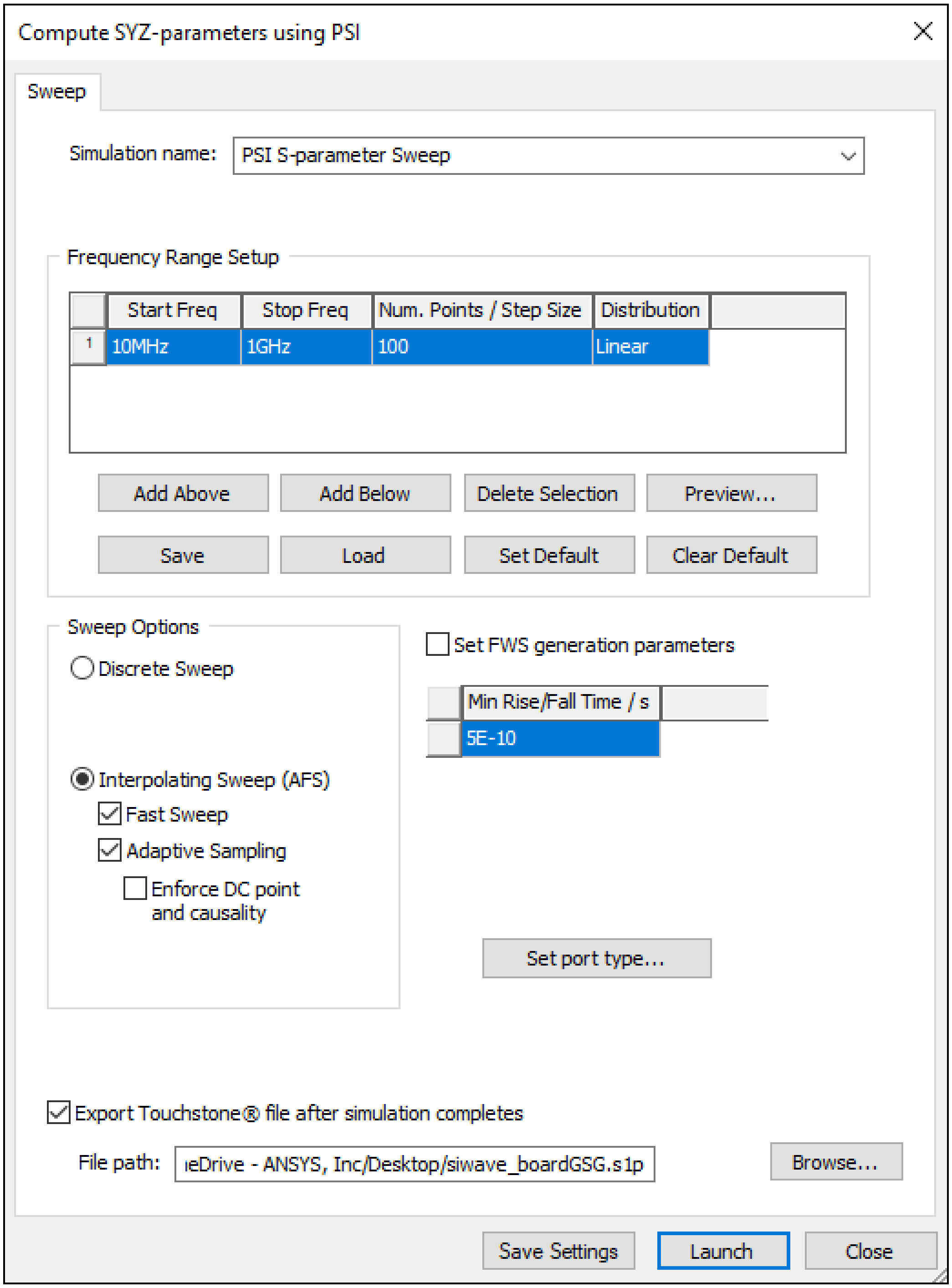
   The **Compute SYZ-parameters using PSI** window opens.



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

memory is available.

Your settings should look like the following:



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.

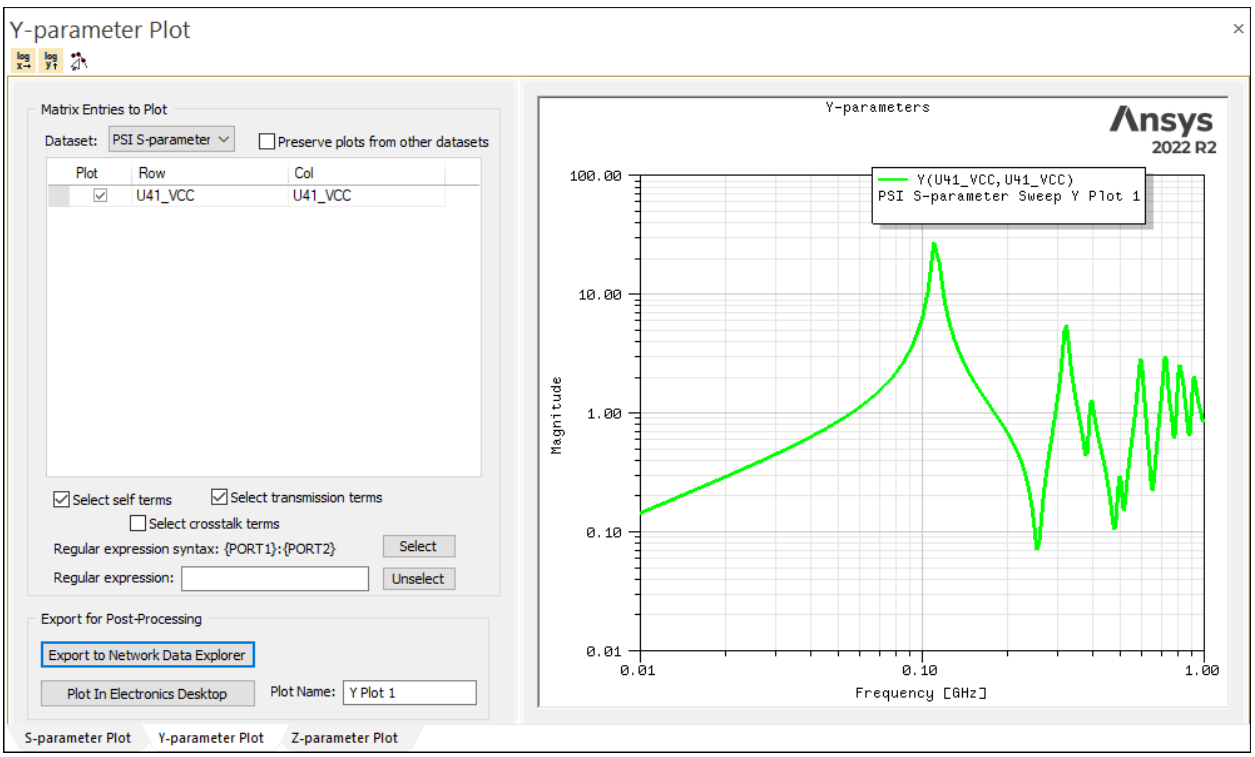
Viewing Impedance Response

To view the frequency-dependent impedance response:

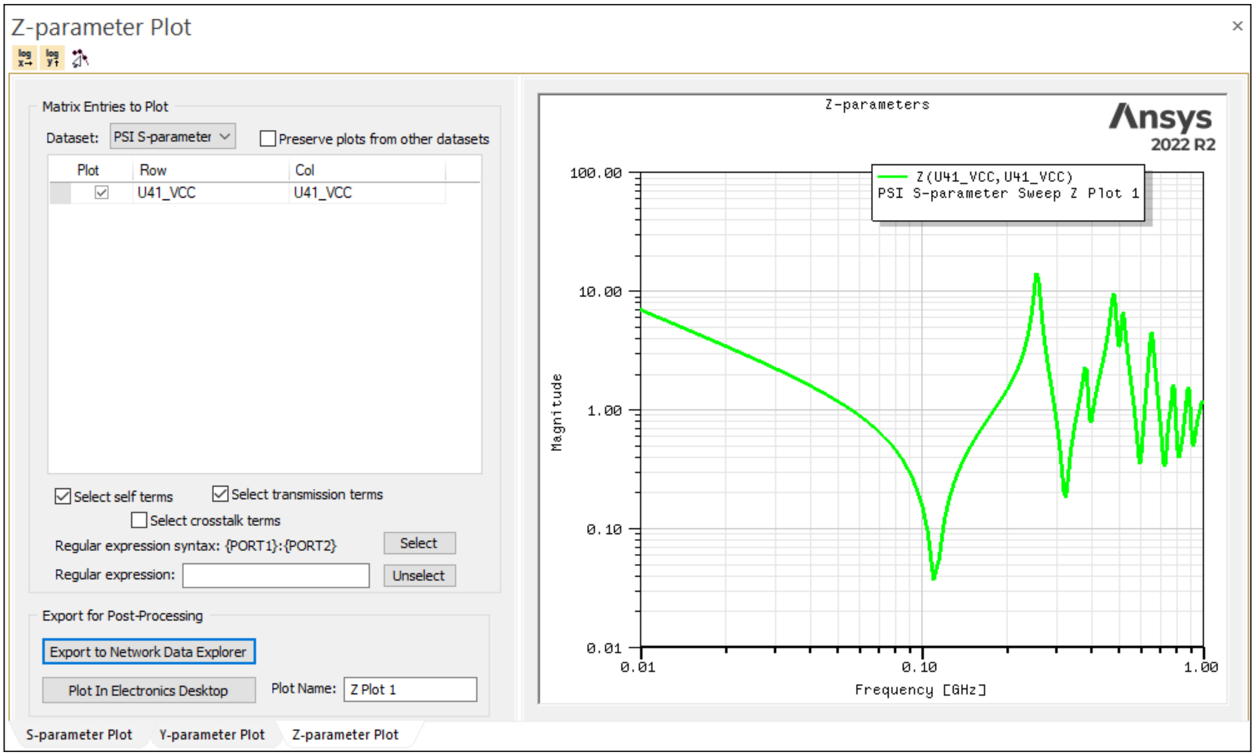
1. From the **Results** workspace, double-click **PSI S-parameter Sweep**.

   > **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.



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



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

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.

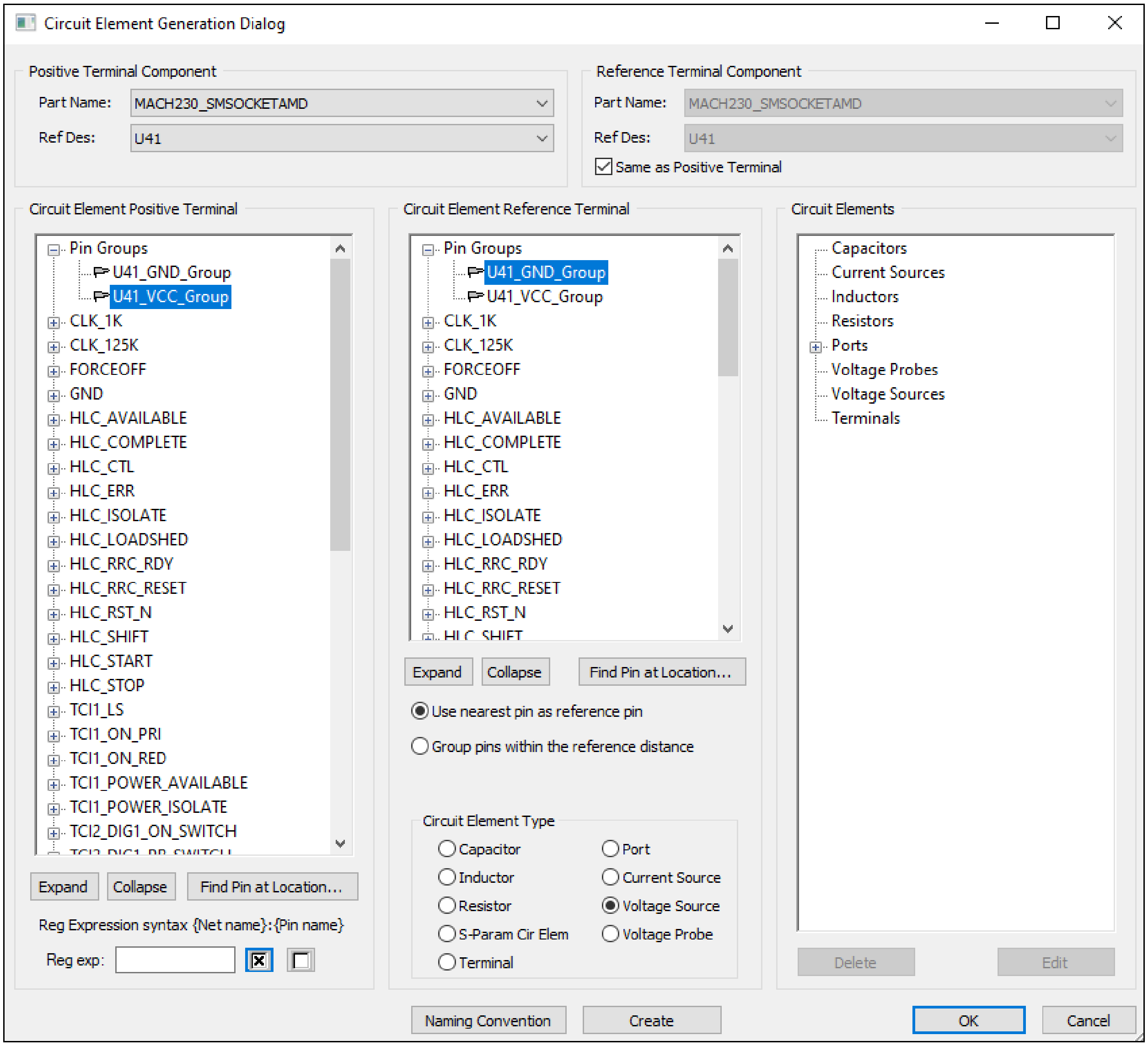
Because the analysis result is a current, the effect of the changing plane impedance will be most clear if a constant voltage excitation is used.

To create a voltage source:

1. Click **Tools** > **Generate Circuit Element on Components**.

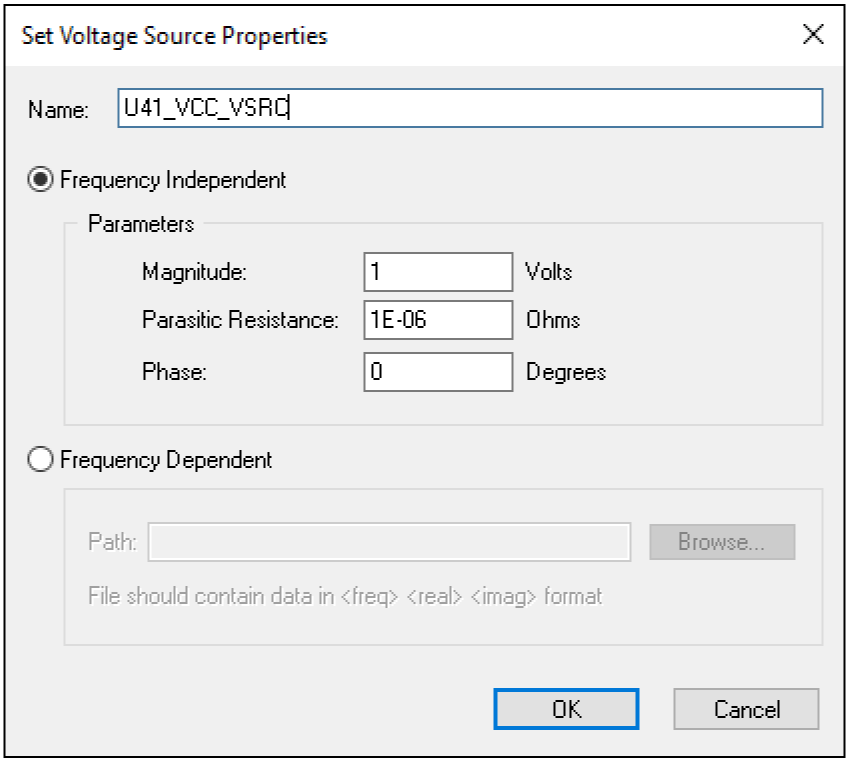
   The **Circuit Element Generation** window appears.
2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
3. In the **Reference Terminal Component** section, check the **Same as Positive 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:



7. Click **Create**.

   The **Set Voltage Source Properties** window appears.



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.

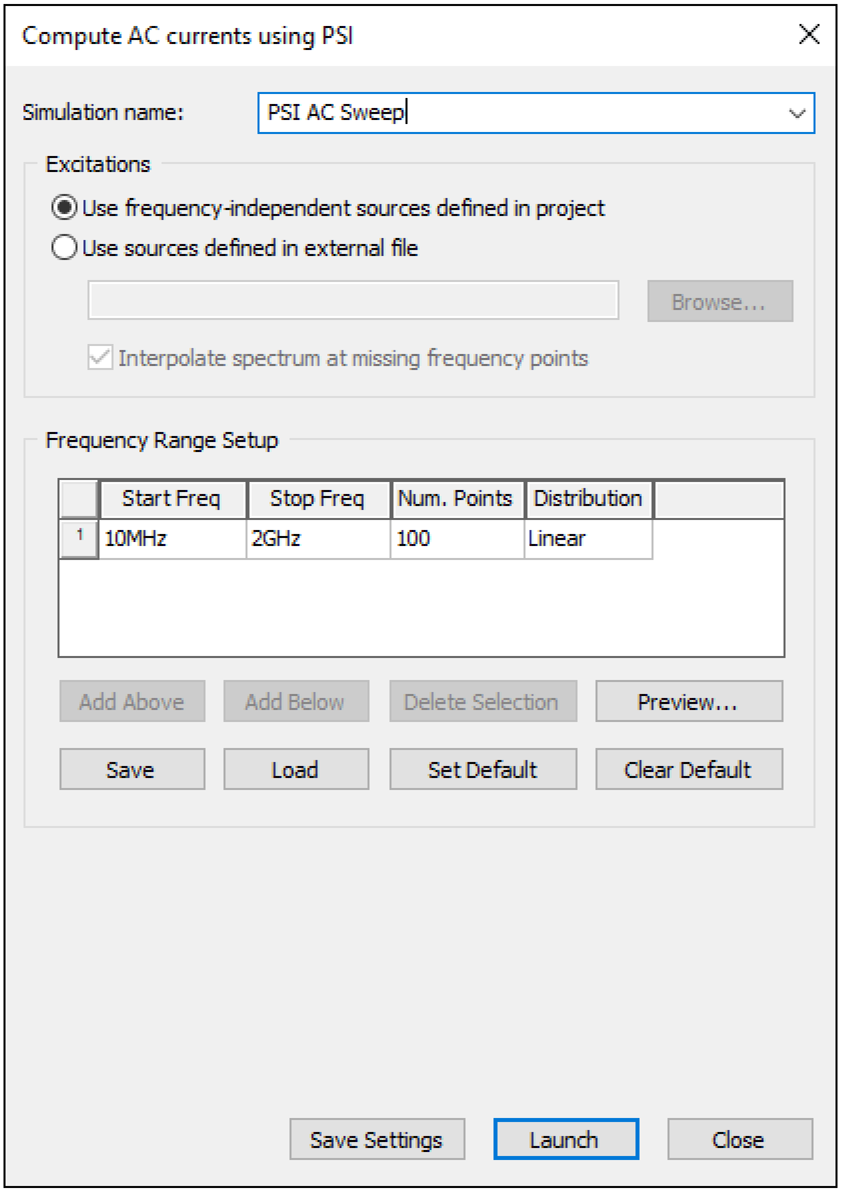
Calculating AC Currents

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.



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:

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

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

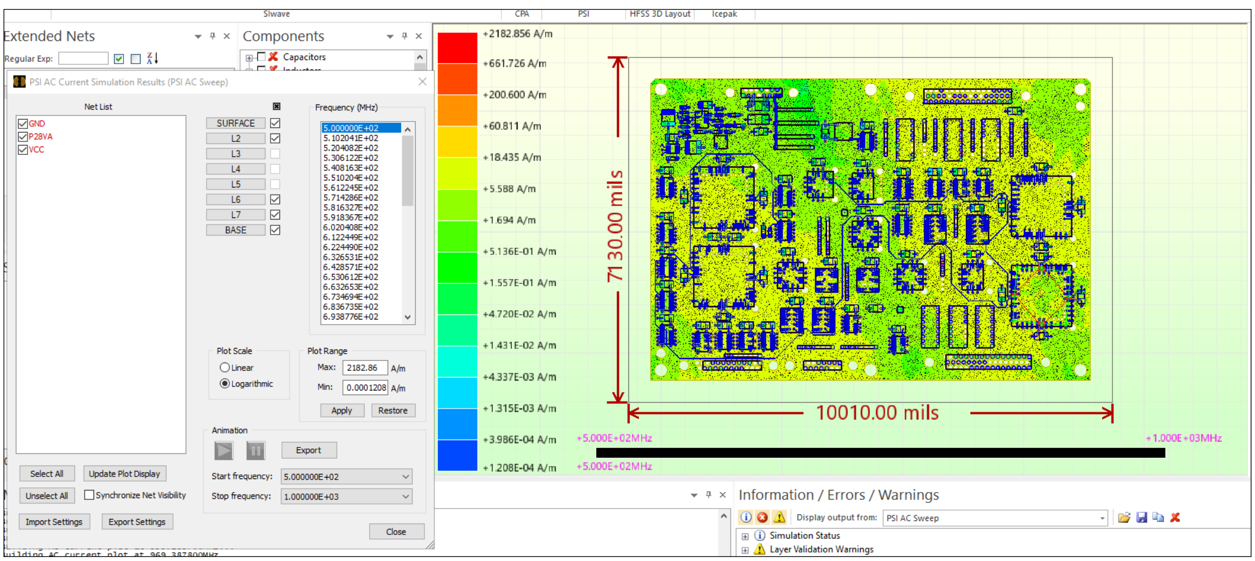
   The **Messages** workspace updates to display a progress bar. When the analysis has completed, you can view AC currents as 2D plots.

Viewing AC Currents as 2D Plots

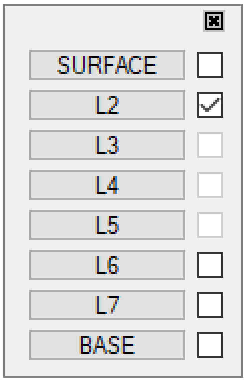
To view the results plot:

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:



2. In the Layer Toggle column (☒), clear the check boxes for every layer *except* **L2**.



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.

3. View the saved file in any text editor.

```
"Freq [Hz]","RadiatedPower [W]"
500000000,0.0564708489195
510204081.633,0.0316060131002
520408163.265,0.0226131402443
530612244.898,0.0238098289766
540816326.531,0.0295210137185
551020408.163,0.0406286155911
561224489.796,0.0626586399411
571428571.429,0.113268233374
581632653.061,0.251048329572
591836734.694,0.47488819016
602040816.327,0.283155981856
612244897.959,0.128254210618
622448979.592,0.0745450042714
632653061.224,0.0544804305976
642857142.857,0.0473802933887
653061224.49,0.0470300456516
663265306.122,0.0519442862736
673469387.755,0.062977816459
683673469.388,0.0837617742927
693877551.02,0.123826913399
704081632.653,0.208039800654
714285714.286,0.384086567773
724489795.918,0.506413955867
734693877.551,0.297220871799
744897959.184,0.164312436241
755102040.816,0.135141364912
765306122.449,0.105237767082
775510204.082,0.0892679461732
785714285.714,0.114190590754
795918367.347,0.201450244492
806122448.98,0.360715624804
816326530.612,0.357135076406
826530612.245,0.250732149771
836734693.878,0.175404953492
846938775.51,0.112408788204
857142857.143,0.0831272227951
867346938.776,0.0767627795494
877551020.408,0.0894823899409
887755102.041,0.129584586183
897959183.673,0.213723150777
908163265.306,0.297288757457
918367346.939,0.243862625862
928571428.571,0.150671654475
938775510.204,0.0990916886915
948979591.837,0.0786709032629
959183673.469,0.0663596901453
969387755.102,0.054406081455
979591836.735,0.0496260801285
989795918.367,0.055217605595
```

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

7 - Frequency Sweep of Voltages

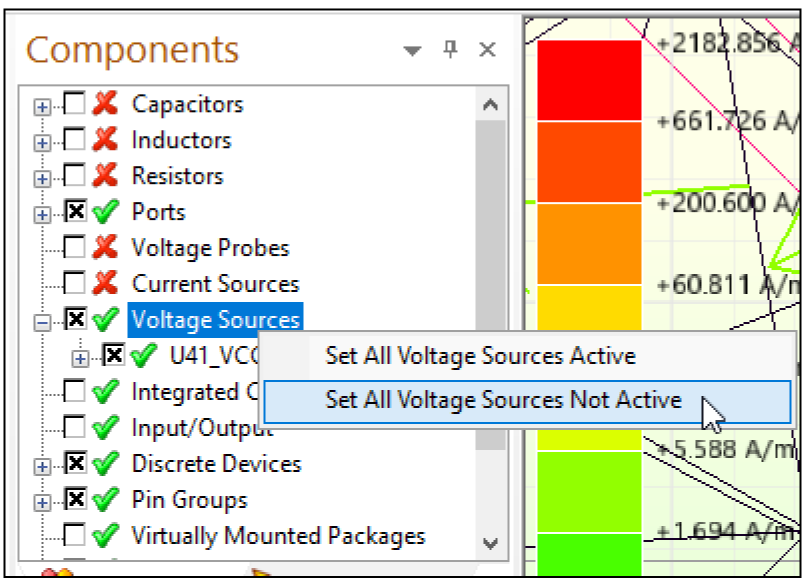
This section explains how to perform the following tasks:

- 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

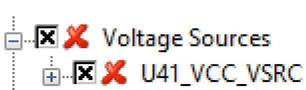
Disabling Voltage Sources

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.



Creating a Current Source on a Component

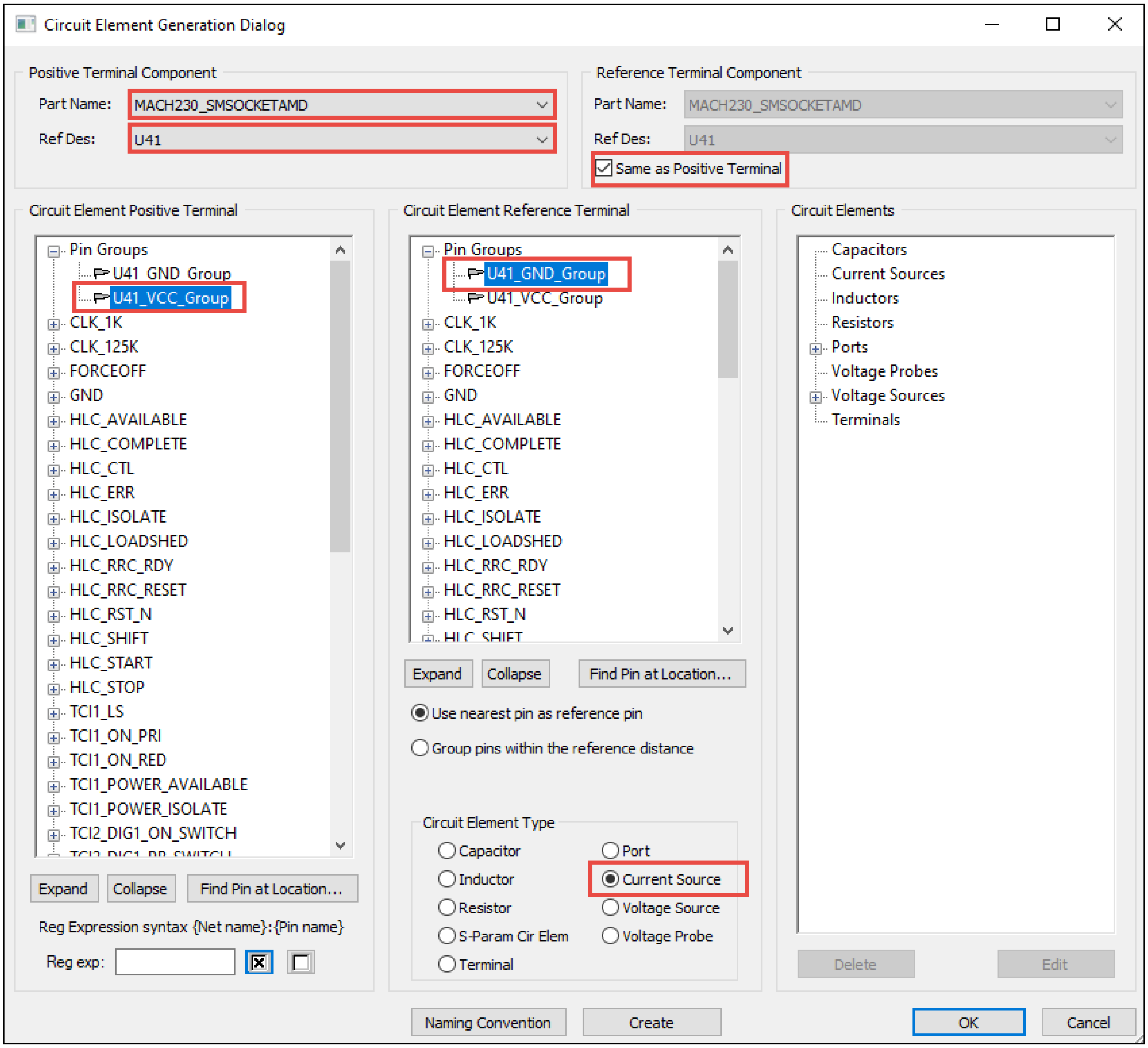
The Frequency Sweep analysis requires one or more sources as excitations.

To define a Current Source:

1. Click **Tools** > **Generate Circuit Element on Components**.

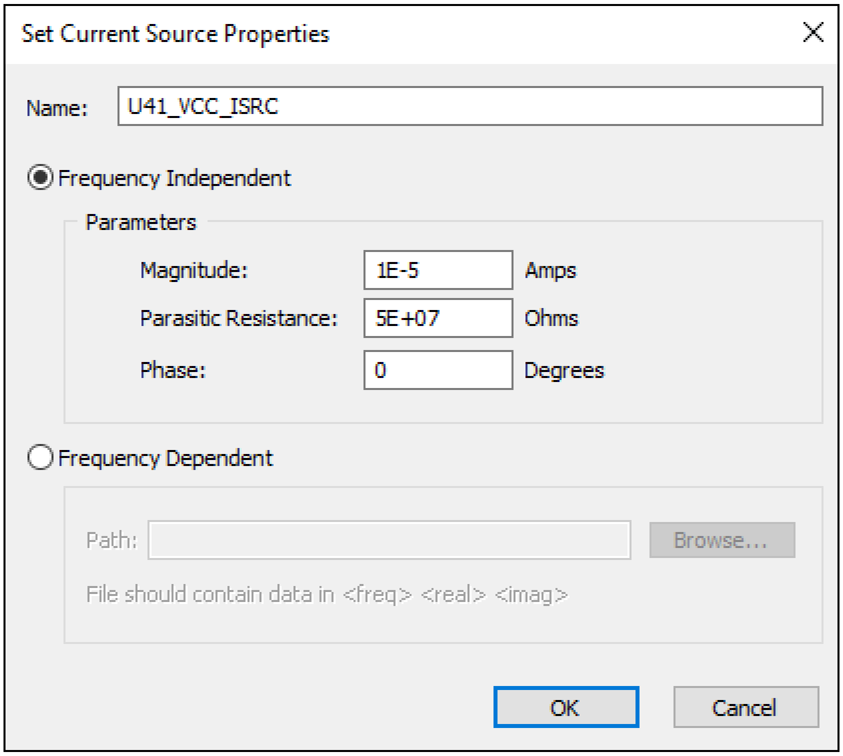
   The **Circuit Element Generation** window appears.
2. In the **Positive Terminal Component** section, use the **Part Name** drop-down menu to select **MACH230_SMSOCKETAMD** and the **Ref Des** drop-down menu to select **U41**.
3. In the **Reference Terminal Component** section, select **Same as Positive 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:



7. Click **Create** .

   The **Set Current Source Properties** window appears.



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

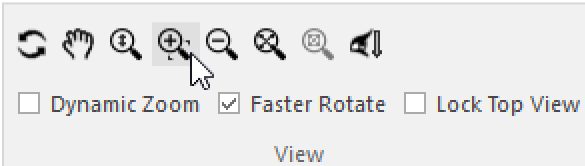
The problem of induced ripple voltage in the Power Distribution Network (PDN) can be investigated by measuring the voltage between the VCC and GND nets, using a voltage probe placed at a location of interest on the PCB.

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.

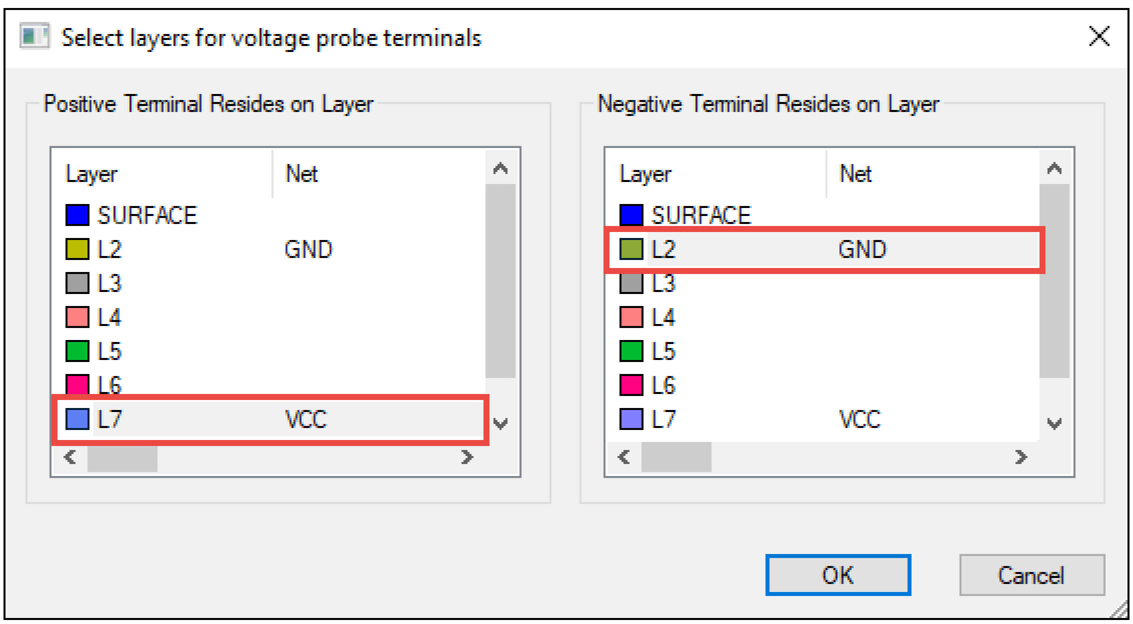
2. Select the **View** tab. Use the **Zoom In** icon (or your mouse wheel) to enlarge the lower-right corner of the PCB.



3. Select the **Home** tab.
4. In the **Circuit Elements** area, select the **Add Voltage Probe** icon ( ).

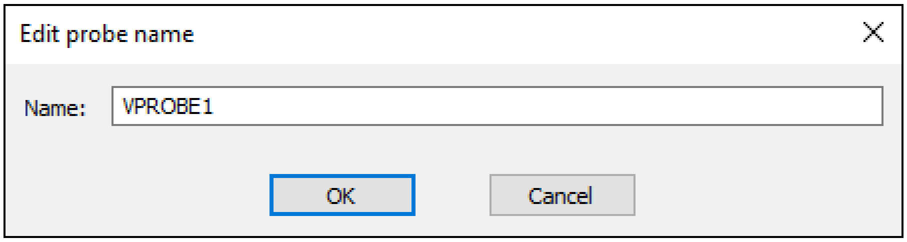
   The cursor changes to a small bullseye.
5. Use the location indicators at the bottom-right of the screen to position your cursor at x: 8500 mils, y: 0 mils.
6. Click the left mouse button twice, slowly each time. The first click locates the positive terminal, the second click is for the reference terminal.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

   The **Select layers for voltage probe terminals** window appears.



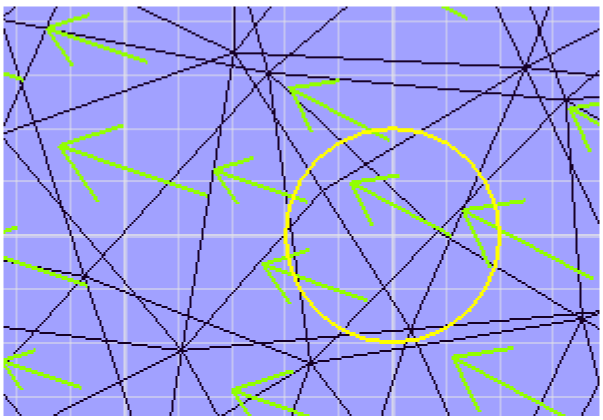
7. For the **Positive Terminal Resides on Layer**, select **L7 (VCC)**.
8. For the **Negative Terminal Resides on Layer**, select **L2 (GND)**.
9. Click **OK** to close the **Select layers for voltage probe terminals** window.

   The **Edit Probe Name** window appears.



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

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



Running a Frequency Sweep

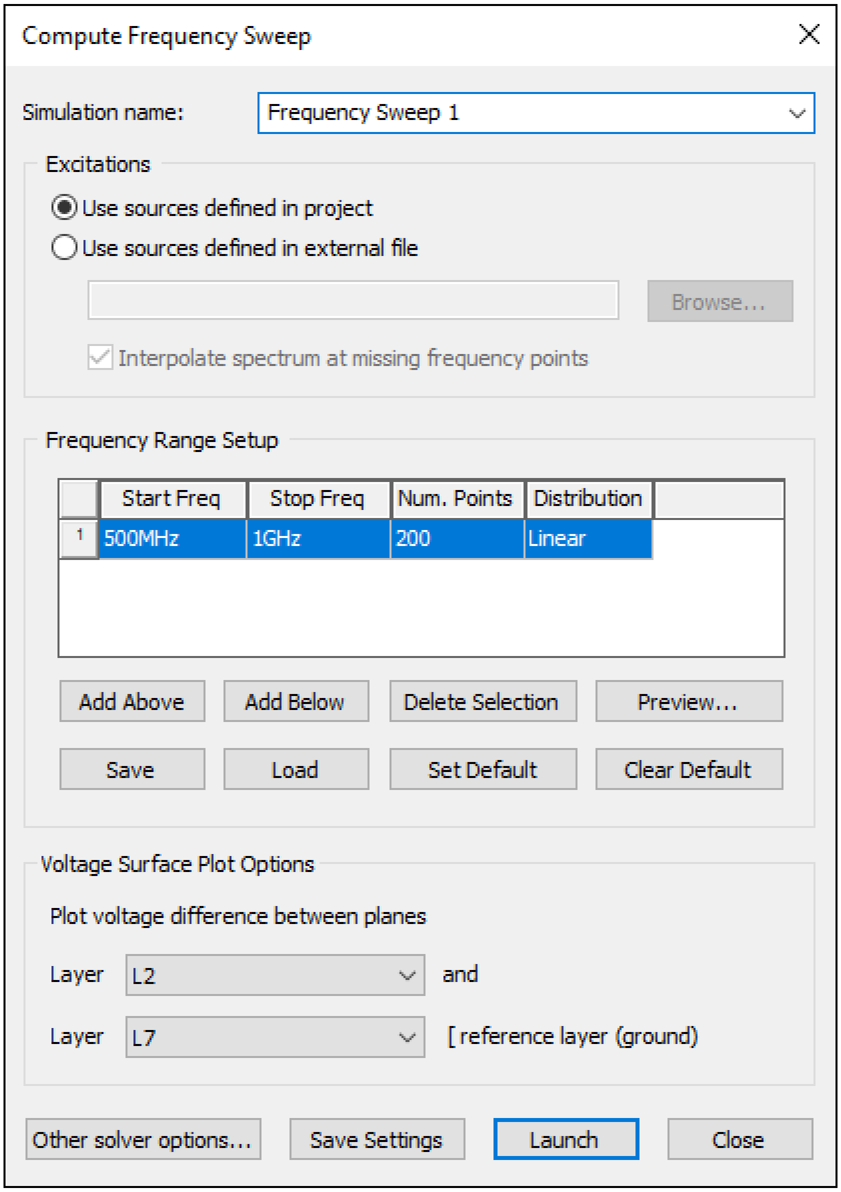
Frequency sweep calculates the voltages across the VCC and GND planes at the locations of the probe. The SIwave frequency sweep shows how the current drawn by a device can lead to voltage ripple in the power distribution system.

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:



8. Click **Launch** to begin the simulation.

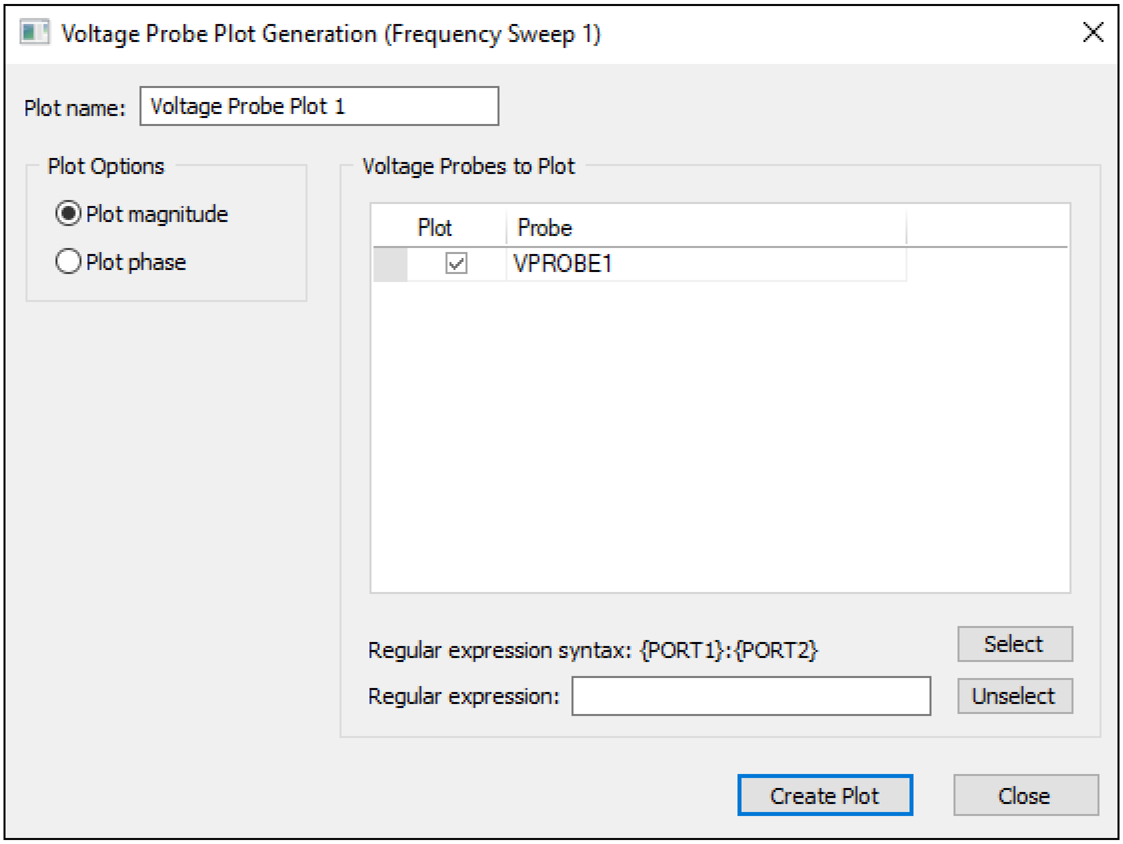
   The **Messages** workspace displays a progress bar that lets you know when the simulation is complete.

Plotting Probe Voltage

To plot the voltages at the probe locations:

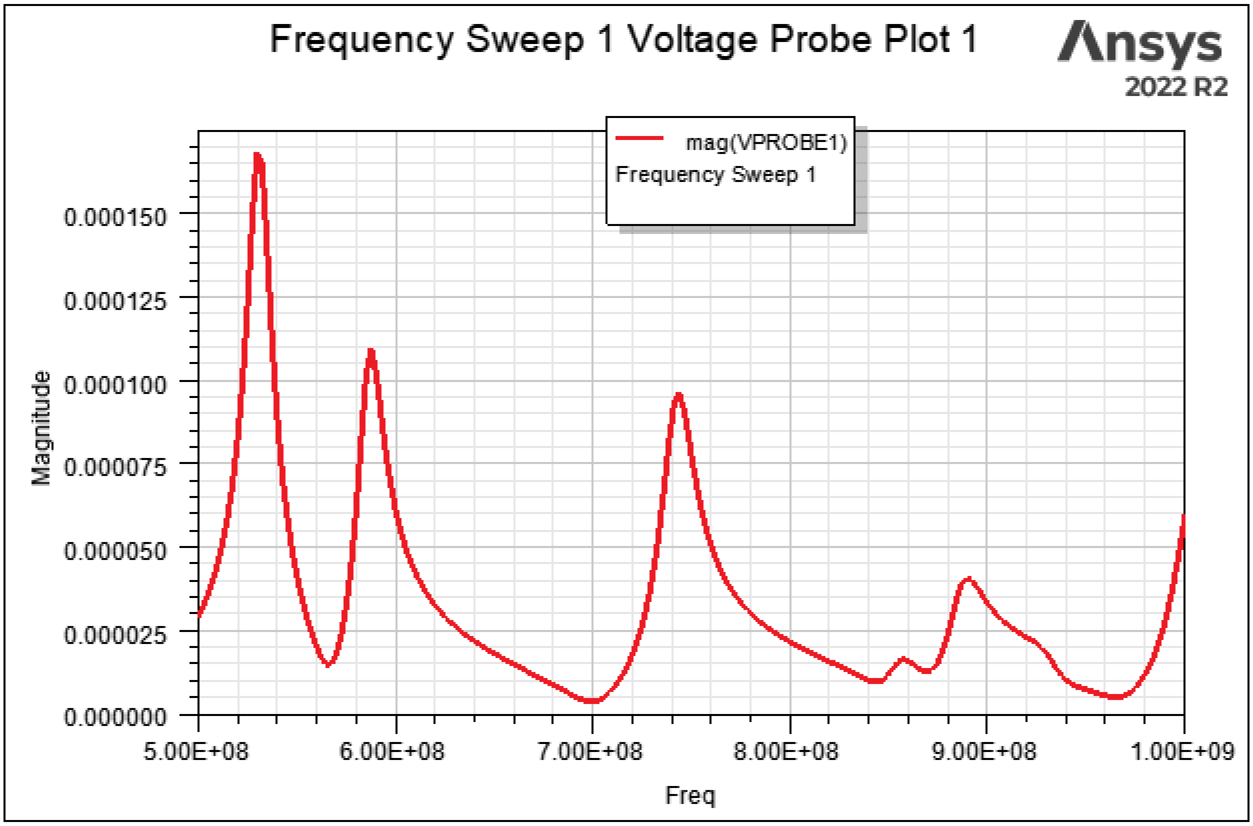
1. Select the **Results** tab.
2. Click **Frequency Sweep** > **Frequency Sweep 1 Results** > **Plot Probe Voltages**.

   The **Voltage Probe Plot Generation** window appears.



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.