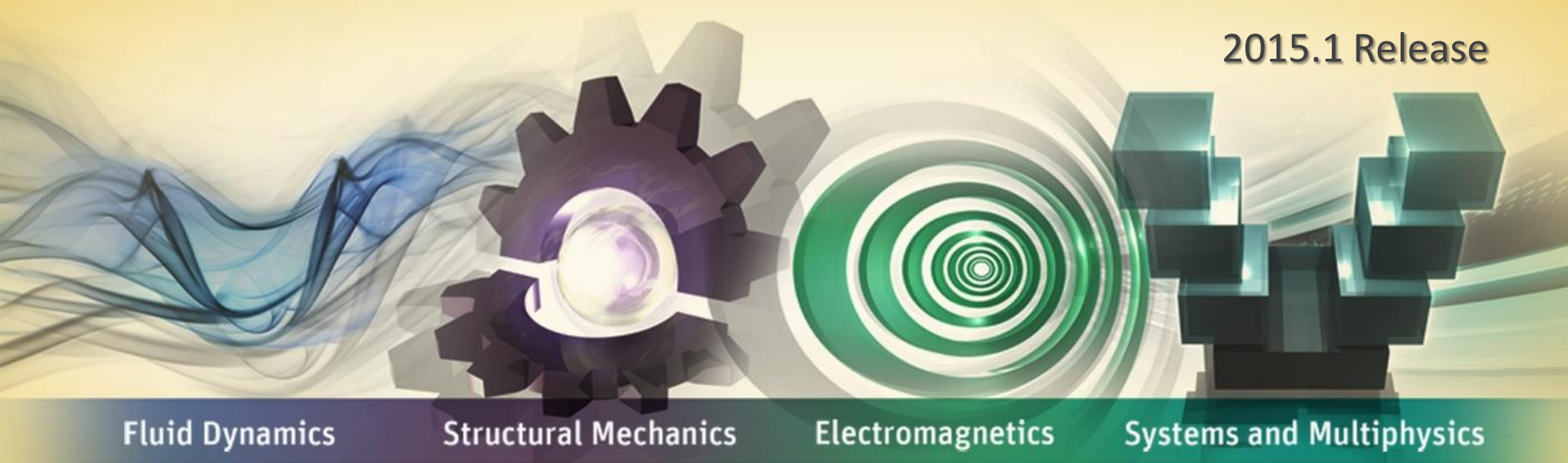


Workshop 1-2: Impedance

2015.1 Release



Fluid Dynamics

Structural Mechanics

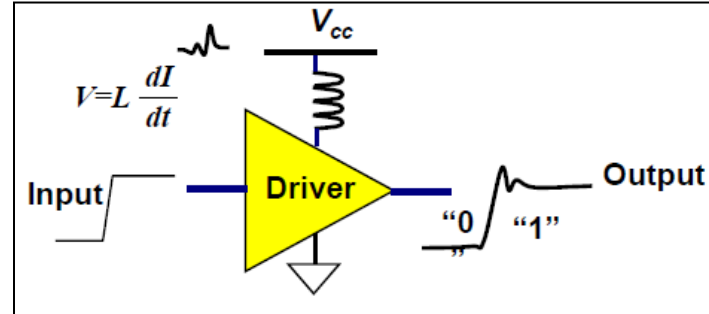
Electromagnetics

Systems and Multiphysics

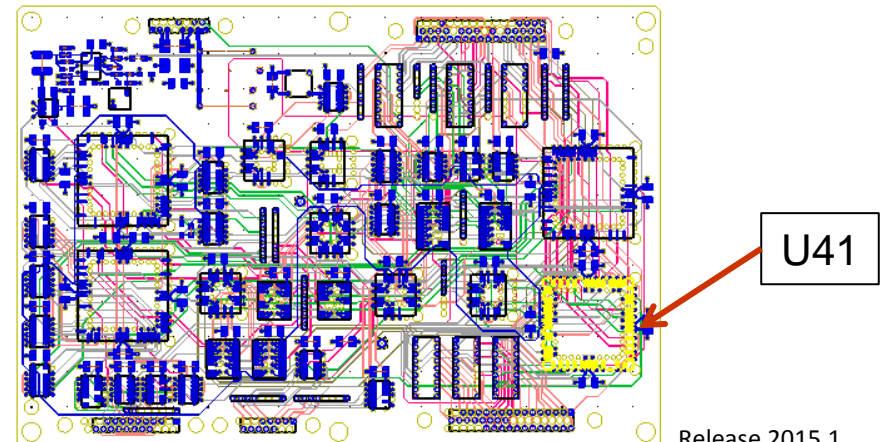
Introduction to ANSYS SIwave

- **Impedance Response of the Power Distribution System**

- The first part of this exercise described how to import a board design and calculate the resonant modes. The amplitude of one resonant mode was reduced by strategically placing decoupling capacitors between the VCC and ground nets.
- The active components on the PCB draw current through the net VCC (or other power supply nets). If the impedance of the VCC net is too large, then ripple voltage is induced when components switch. The ripple voltage may interfere with other devices. For example, the “0” – “1” transition shown below induces ripple voltage between the VCC net and ground. In this case, the inductance (e.g. impedance) of the VCC net connection causes induced ripple voltage.



- The goal of the following exercise is to calculate the frequency dependent impedance of the power supply net VCC as seen from the component U41. It will be shown that the frequencies determined by the resonant mode analysis correspond to peaks in the impedance of the power supply (VCC) net.

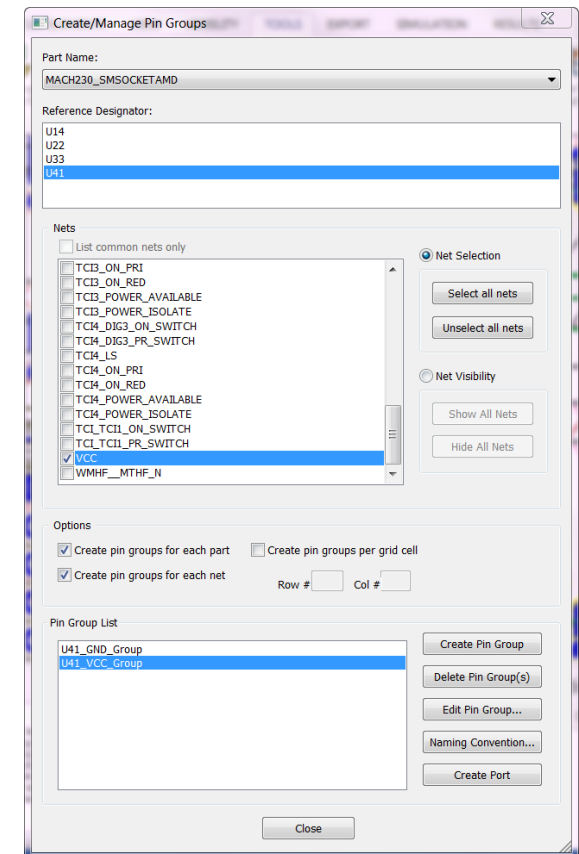
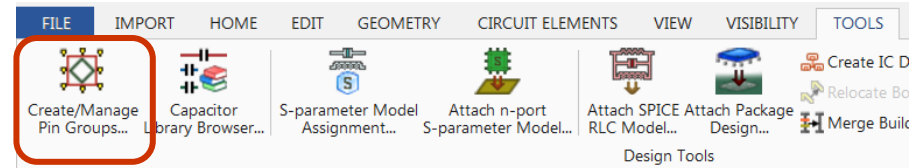


Example – Impedance Response

- Use the *siwave_board.siw* that was created in Workshop 1.1

- **Define Pin Groups**

- Launch Create/Manage Pin Groups from the **Tools** tab
- Select the component
 - Part Name: **MACH230_SMSOCKETAMD**
 - Refdes: **U41**
- From Nets:
 - Select Nets: **GND and VCC**
 - Select all pins using the **Select All Pins** toolbar
- Options:
 - Click the **Create Pin Groups for each part**
 - Create pin groups for each net
- Click the Create Pin Group button
- In the Pin Group List there should be one GND_Group and one VCC_group
- Click **Close**



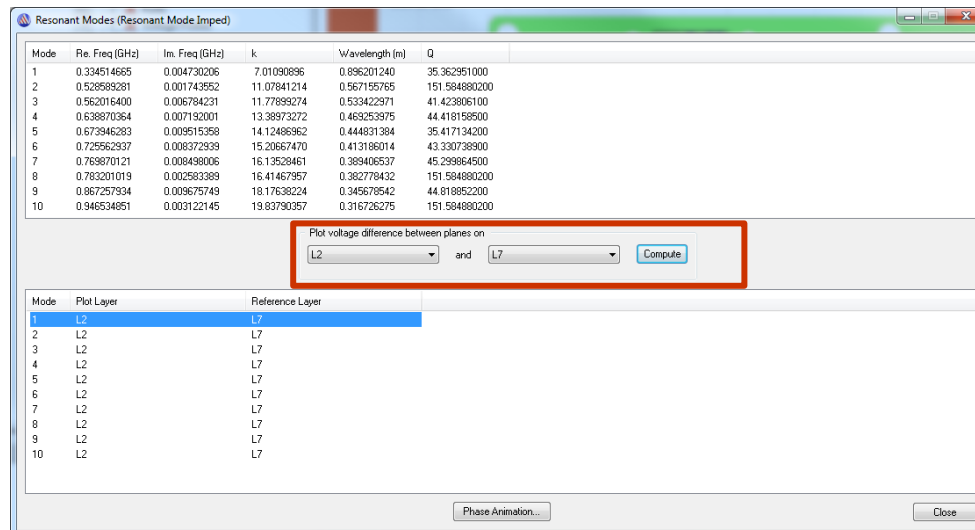
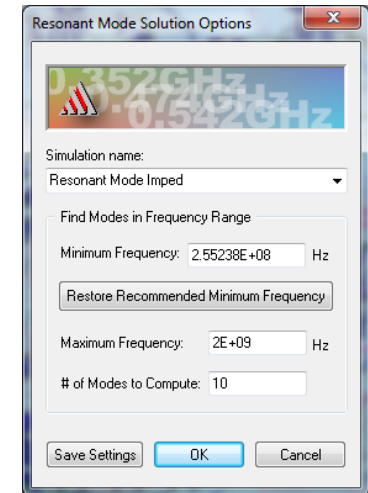
Example – Impedance Response

• Resonant Mode Analysis

- Select the menu item **Simulation > Slwave > Compute Resonant Modes**
 - Simulation name: **Resonant Mode Imped**
 - Minimum Frequency: **2.55238E+08**
 - Maximum Frequency: **2E+09**
 - # of Modes to Compute: **10**
 - Click the **Launch** button

• Viewing Results

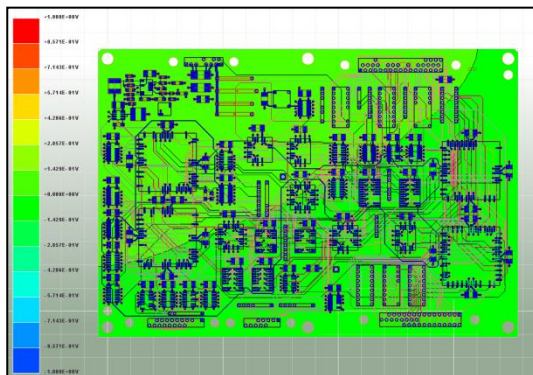
- Select the menu item **Results > Slwave > Resonant Modes > Resonant Mode Imped > View Results.**
 - Generate the voltage plot between planes **L2** and **L7** by selecting the layers and pressing **Compute**. This computation is required to generate the 2D plot from existing solution data.



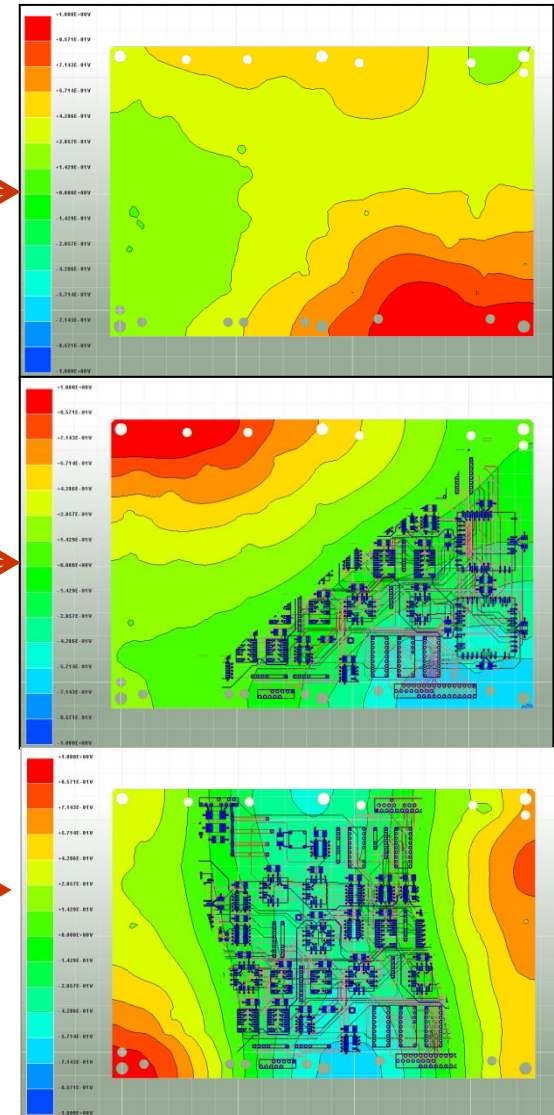
Example – Impedance Response

• Resonant Mode Analysis

- Note that some of the resonances have high amplitude in the lower right corner of the board. The voltage distribution for some of the resonant modes are shown below.
- Based on the resonant mode analysis, the impedance is expected to exhibit peaks at 628 MHz, 685 MHz and 919 MHz and other higher frequencies. The resonance at 325 MHz has very low amplitude at U41 and therefore should not have a significant impact on the impedance..

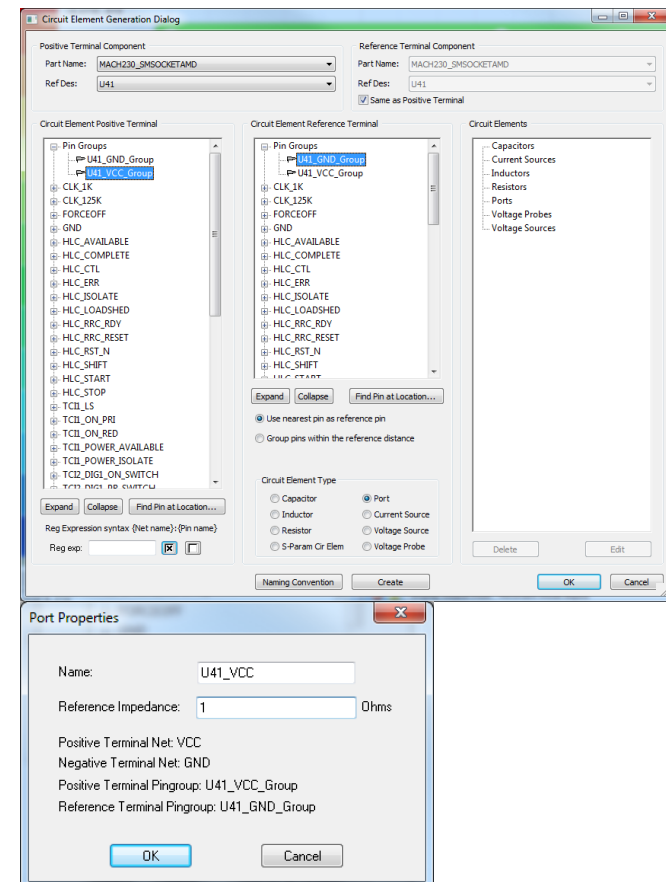
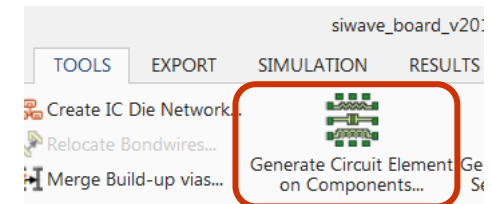


Mode	Re. Freq (GHz)
1	0.325134262
2	0.628362767
3	0.648778991
4	0.685171117
5	0.770031591
6	0.819851068
7	0.919269607
8	0.984162364
9	1.030375791
10	1.088003913



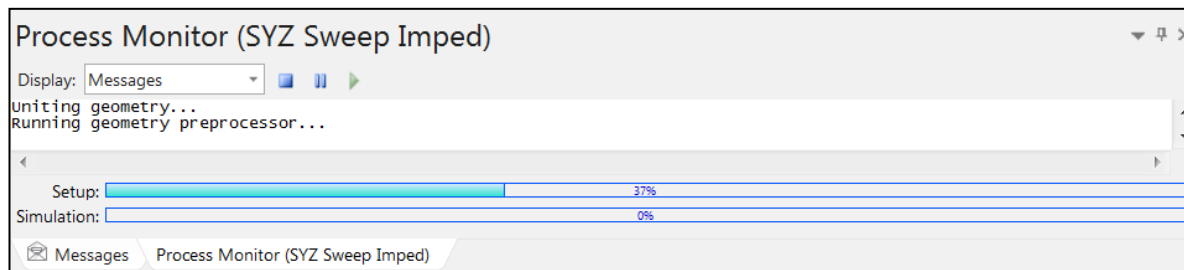
Example – Impedance Response

- **Define the Port between the VCC and GND on U41.**
 - From the Tools tab click on Generate Circuit Elements on Components
 - Positive Terminal Component
 - Part Name: **MACH230_SMSOCKETAMD**
 - Ref Des: **U41**
 - Reference Terminal Component (or **Same as Positive Terminal** ☒ **Checked**)
 - Part Name: **MACH230_SMSOCKETAMD**
 - Ref Des: **U41**
 - In Circuit Element Positive Terminal expand the Pin Groups item.
 - Choose the pin group **U41_VCC_Group**
 - In Circuit Element Reference Terminal expand the Pin Groups item.
 - Choose the pin group **U41_GND_Group**
 - Under Circuit Element Type make sure **Port** is selected.
 - Press the **Create** button.
 - A dialog window appears asking for the name of the port.
 - Name: **U41_VCC**
 - Reference Impedance: **1 ohm**
 - Click the **OK** button to accept the port definition
 - Click the **OK** button to exit the circuit element generation dialog



Example – Impedance Response

- **Generate Impedance Response at port U41_VCC**
 - From the Simulation Tab click the Compute SYZ Parameters...
 - Simulation Name: **SYZ Sweep Imped**
 - Start Freq/Hz: **100 MHz**
 - Stop Freq/Hz: **1GHz**
 - Num Points: **1000**
 - Distribution: **By Decade**
 - Sweep Selection:
 - **Interpolating Sweep**
 - Error Tolerance: **0.001**
 - Click the **Launch** button
 - The solution process is monitored in the progress window as shown here



- **Viewing results**
 - After the simulation completes
 - Select the **Results Tab > SYZ > SYZ Sweep Imped > Plot Magnitude/Phase**

Compute SYZ-parameters

Sweep Sensitivity Distributed Analysis (HPC)

Simulation name: SYZ Sweep Imped

☐ Compute exact DC point

Frequency Range Setup

	Start Freq	Stop Freq	Num. Points / Step Size	Distribution
1	100MHz	1GHz	1000	By Decade

Add Above Add Below Delete Selection Preview...

Sweep Selection

☐ Discrete Sweep

☒ Interpolating Sweep

Relative error for S: 0.001

☐ Set FWS generation parameters

Min Rise/Fall Time / s: 1E-10

☐ SIwave with 3D DDM

Other solver options...

☒ Export Touchstone® file after simulation completes

File path: D:\2014_release_EM\TrainingMaterials\trunk\SIwave Browse...

Save Settings Launch Close

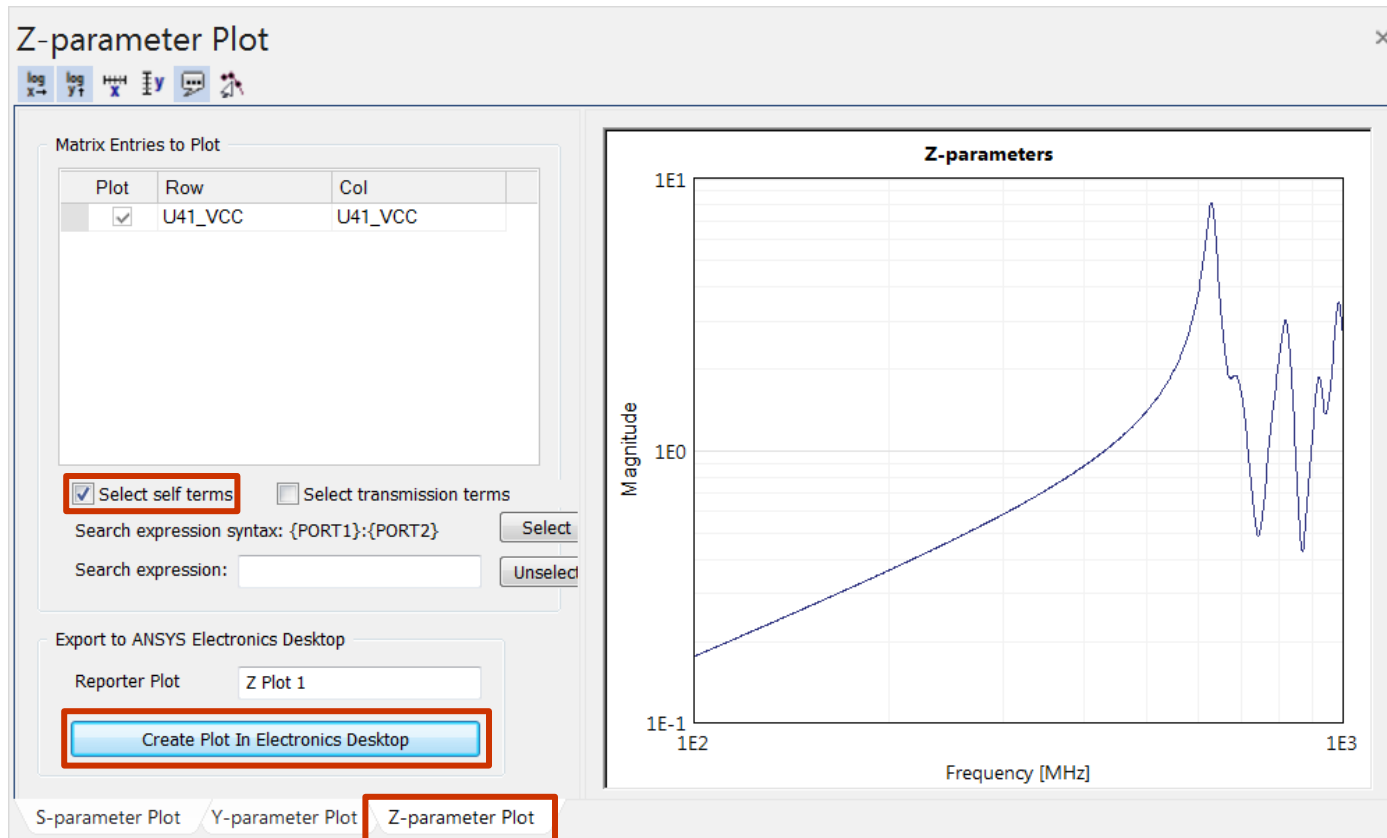
Example – Impedance Response

• Viewing Results

- In the plot generation dialog window choose tab **Z-parameter Plot**.

Note: The model has only one port and therefore only the self terminal will be selected, **Select Self terms** ☒ **Checked**.

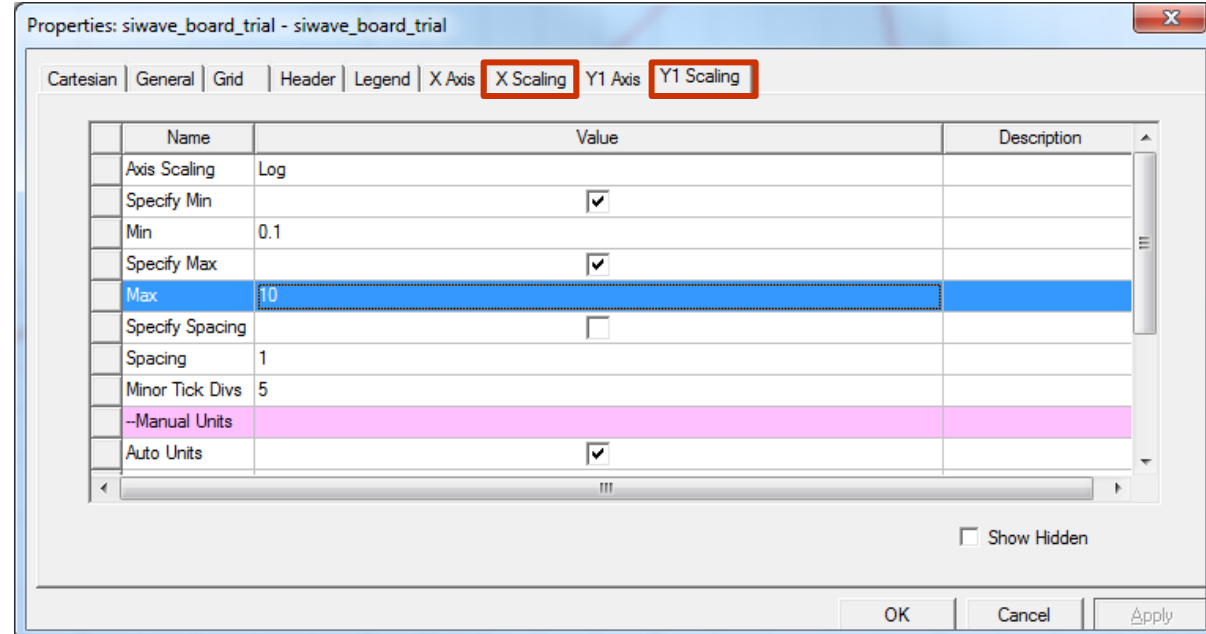
- Click the **Create Plot in Electronics Desktop** button
- The ANSYS Electronics Desktop will be launched and a plot of the impedance magnitude vs. Frequency will be displayed



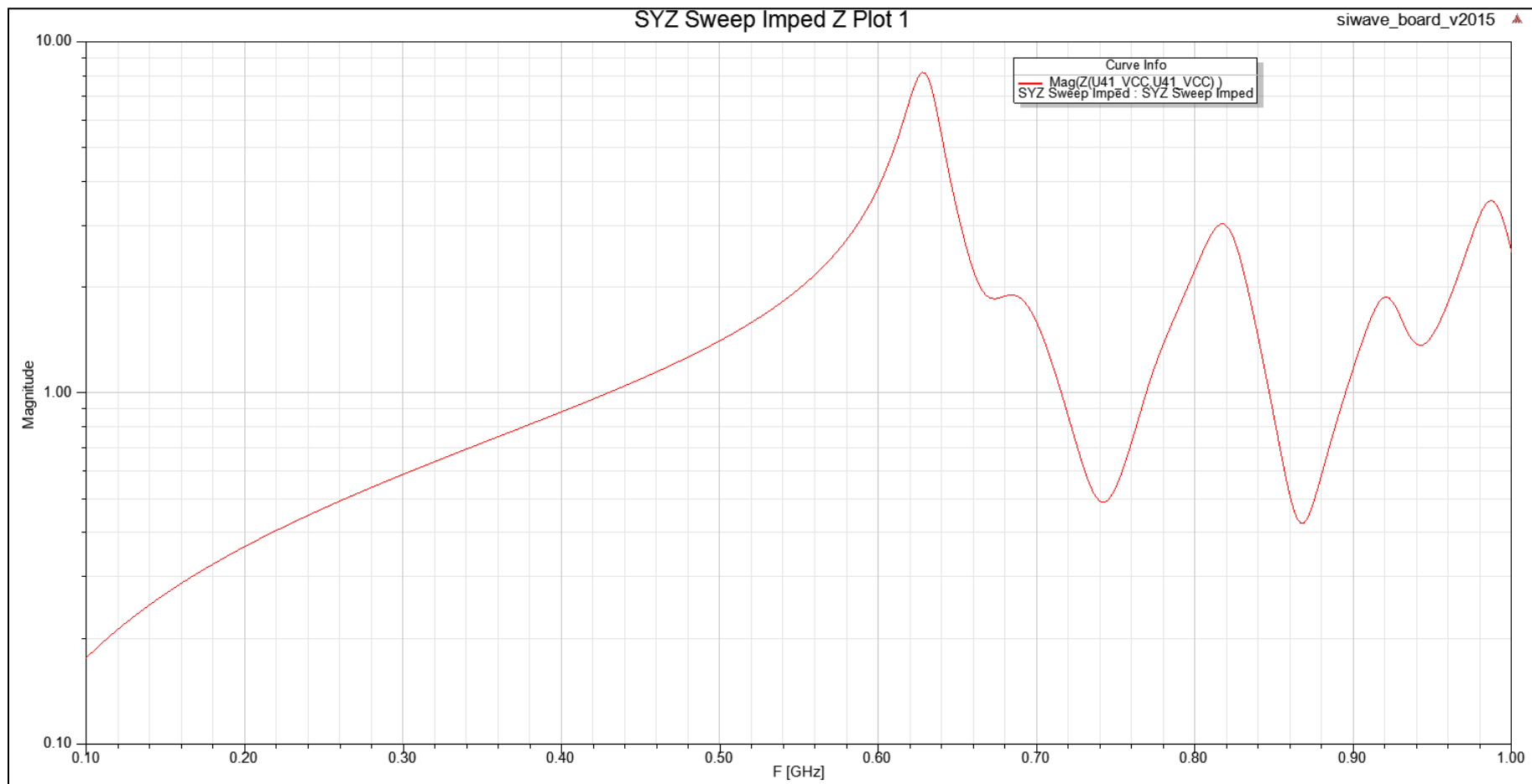
Example – Impedance Response

• Viewing Results

- Change the scale of the vertical axis by double-clicking on the vertical (Y) axis labels. The following dialog window appears.
 - Select the **Y1 Scaling** tab
 - Axis Scaling: **Log**
 - Choose Specify Min: ☒ **Checked**
 - Enter **0.1** for the minimum scale value.
 - Choose Specify Max : ☒ **Checked**
 - Enter **10** for the maximum scale value.
 - Click the **Apply** button
- Change the scale of the horizontal axis selecting the **X Scaling** tab.
 - Select the Scaling tab
 - Axis Scaling: **Linear**
 - Click the **OK** button



- Viewing Results



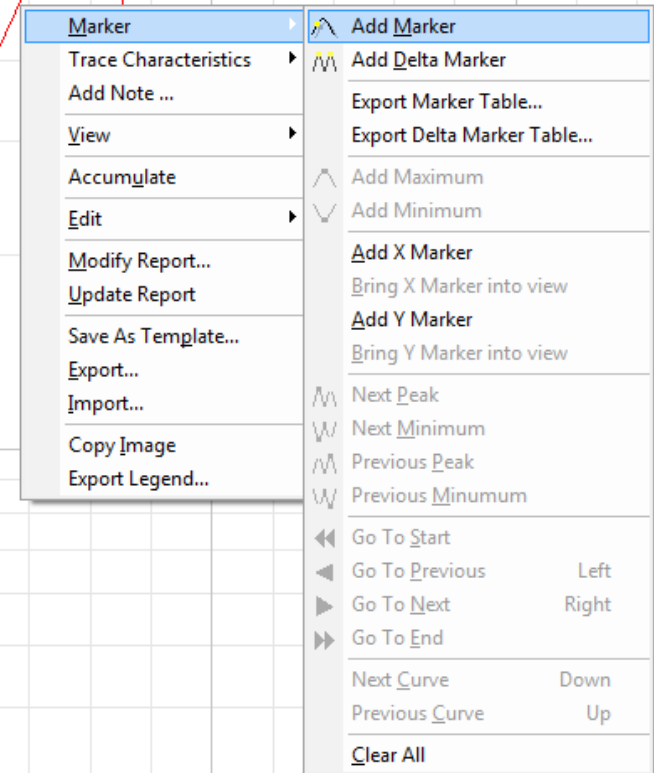
Example – Impedance Response

• Analyze the Results

- On the impedance profile in the plot:
Right Mouse Click > Marker > Add Marker
- Move the mouse to the top of the first peak and **Mouse Left Button** click to add the first marker.
- Continue along the impedance profile and place markers at all of the peaks in the profile. (There should be 5 of them)
- When finished press **Esc** to exit the marker place mode.
- There should now be a legend that maps each of the points to the markers that were placed on the signal.

Note: The markers can be dragged along the signal if they are not in the correct location.

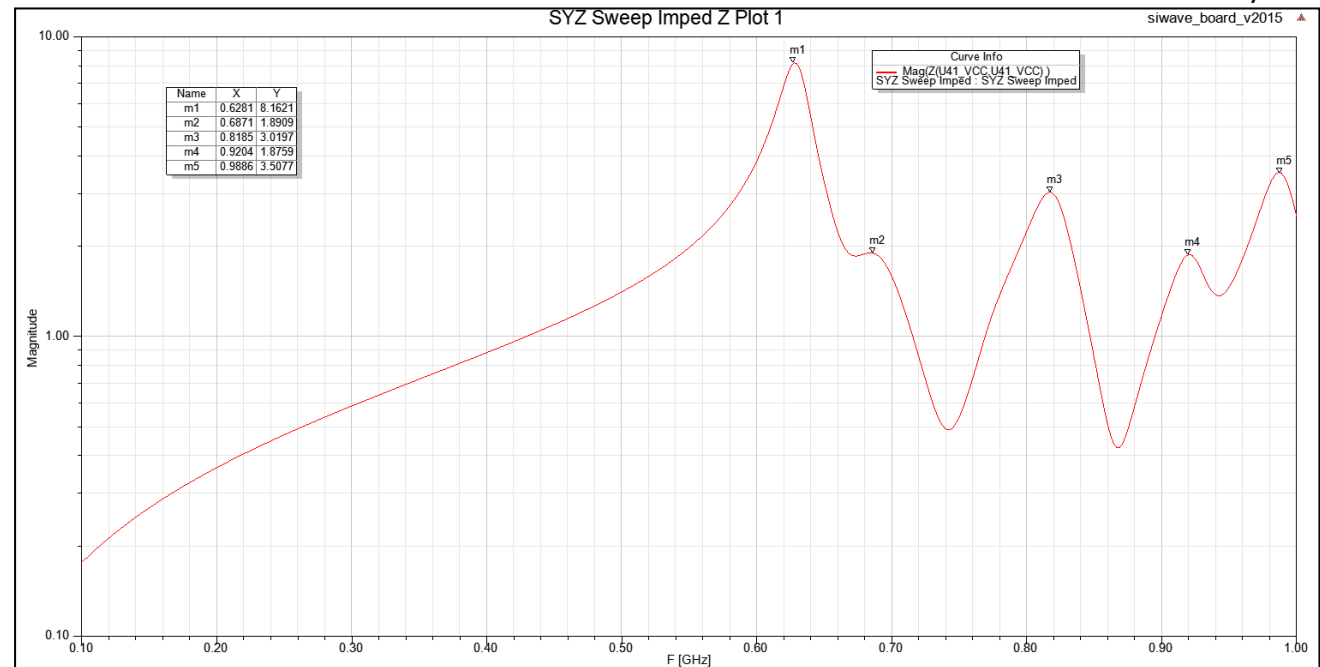
Note: The marker can be forced to the local peak by selecting the marker and then **Right Mouse Click > Marker > Next Peak** or **Previous Peak**.



Example – Impedance Response

- **Impedance Response**

- The frequency dependent impedance shows good correlation with the resonant modes determined using the resonant mode analysis. The plot shown below displays the impedance magnitude. As seen in the resonance analysis the mode at 325 MHz is not seen in the impedance $|Z(f)|$. This observation is expected based on the weak relative amplitude of the corresponding voltage oscillation.
- The peaks at 628 MHz, 819 MHz and 920 MHz correlate with the resonances that were observed earlier in the resonance analysis of the plane.



- **Save the .siw file**

- Select the menu item **File > Save**