Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with HFSS™: Matching Network – Using Tuning in Circuits

Ansys

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 2025 R2
July 2025

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

Conventions Used in this Guide

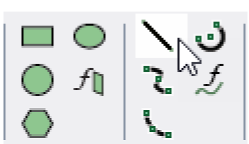
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, "click **HFSS** > **Excitations** > **Assign** > **Port** > **Wave Port**."
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *filename*" means you must type the word **copy**, then type a space, and then type the name of the file.
- 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.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**."



This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  "Right-click and select **Assign Excitation** > **Port** > **Wave Port**" means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

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.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Introduction

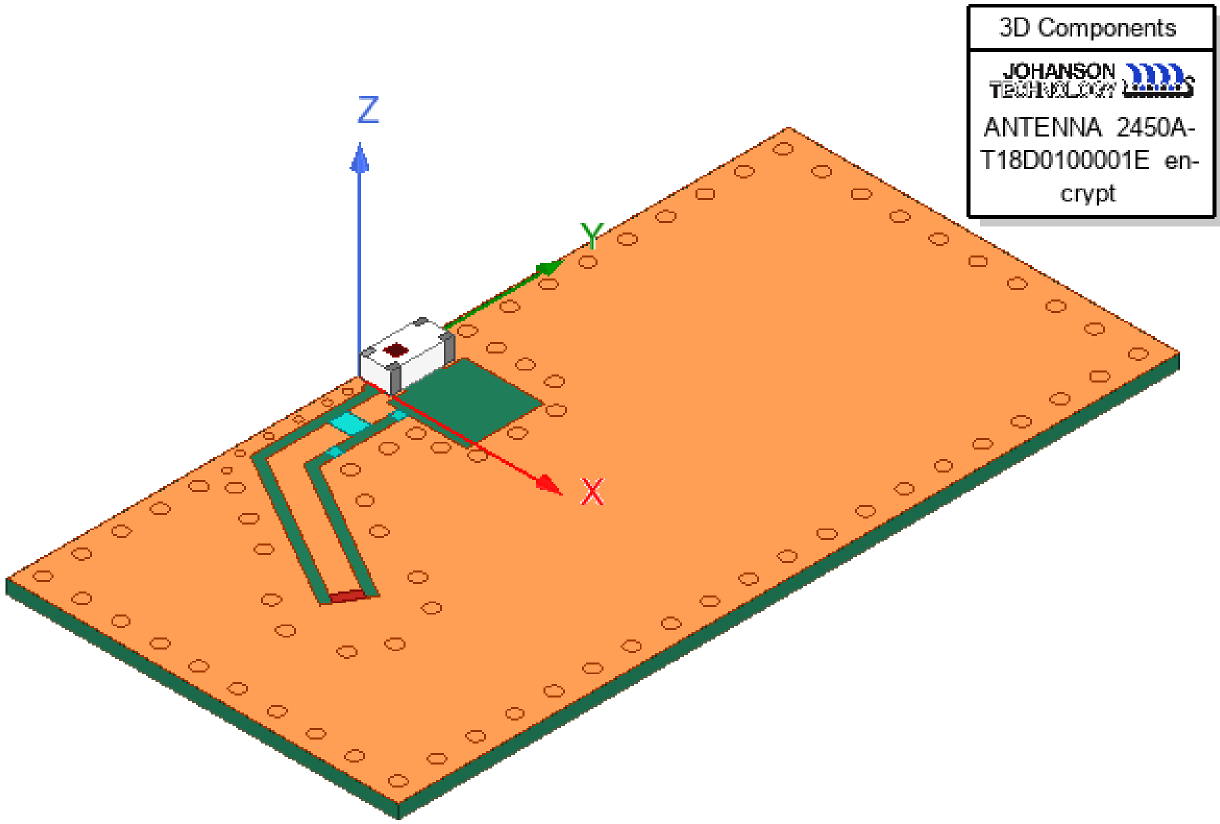
In this *Getting Started* guide, you will learn about dynamic linking capabilities between *HFSS* and *Circuit* designs in the *Ansys Electronics Desktop* application. Specifically, the guide provides an example of a matching network for a Bluetooth chip antenna.

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

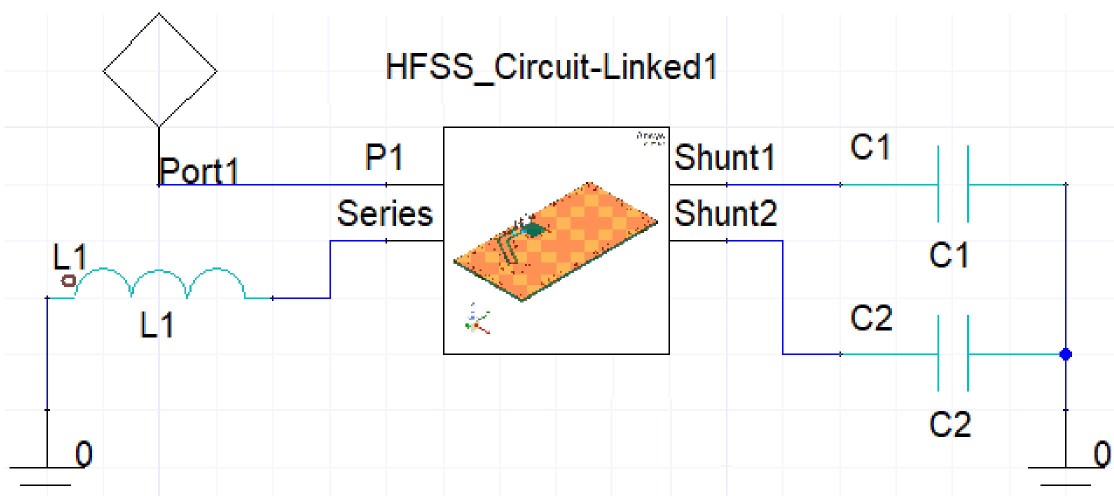
- Dynamically link an HFSS design to a Circuit simulation
- Use the tuning feature in the Circuit design to match the antenna using lumped components (you will tune the component values)
- Create S-parameter plots
- Apply a marker to a plot to assist in component tuning
- Push excitations from the Circuit design to the HFSS design
- Create gain plots (radiation pattern) and overlay them on the model geometry

You will begin with a partially completed project file containing a model of a Johanson evaluation board, to which you will add a Johanson 2450AT18D0100001E Bluetooth chip antenna. There is a completed example model of this evaluation board and antenna included with the Ansys Electronics Desktop installation (in the *Examples\HFSS\Antennas* subfolder). The related example model demonstrates how to automatically optimize the matching network either in a Circuit design linked to the HFSS design or entirely within the HFSS design. However, in this getting started guide you will use an incomplete version of the example model (in the *Help\HFSS* subfolder) as your starting point, and you will learn how to tune the matching network manually.

The antenna is designed to work at a frequency in the range of 2.4 to 2.48 GHz. The following images shows the evaluation board with the chip antenna mounted to it:



The next image shows the linked HFSS evaluation board along with its matching network circuit:



2 - Set Up the Project

In this chapter, you will perform the following tasks:

- Launch the Ansys Electronics Desktop application
- Open the evaluation board project file
- Enable legacy view orientations
- Insert the chip antenna from the *Components Libraries*
- Rename the predefined lumped port excitation

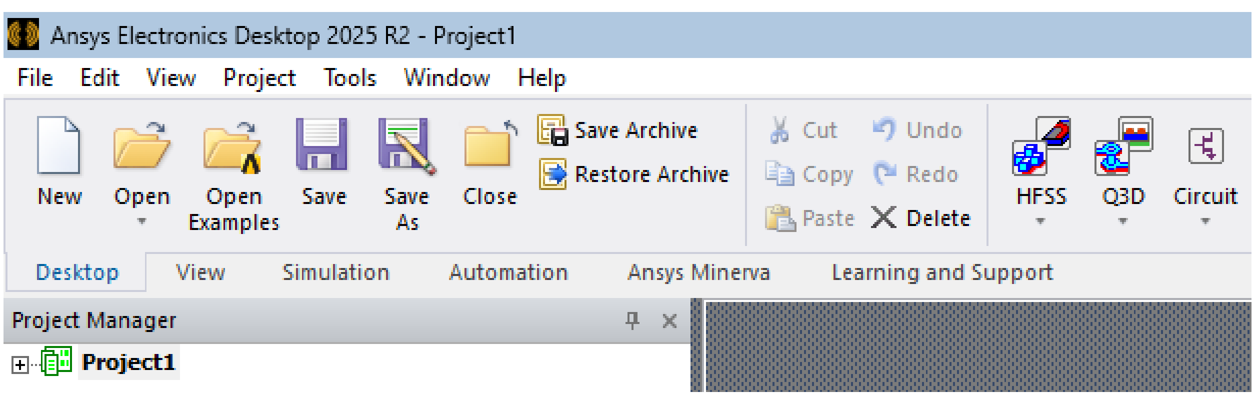
Launch Ansys Electronics Desktop

If the Ansys Electronics Desktop application is not already running, launch it now as follows:

1. Locate **Ansys EM Suite 2025 R2** > **Ansys Electronics Desktop 2025 R2** in your Windows Start menu's All Programs list to launch it.

   Alternatively, click an Ansys EDT shortcut that you have pinned to the Start Menu or Taskbar, or double-click a desktop shortcut you may have added.

   The Ansys Electronics Desktop application opens:



> **Note:**
>
> When you launch the application, a new, blank project is created automatically. For this exercise, you will not start with a new project. Therefore, you will close it in the next step.

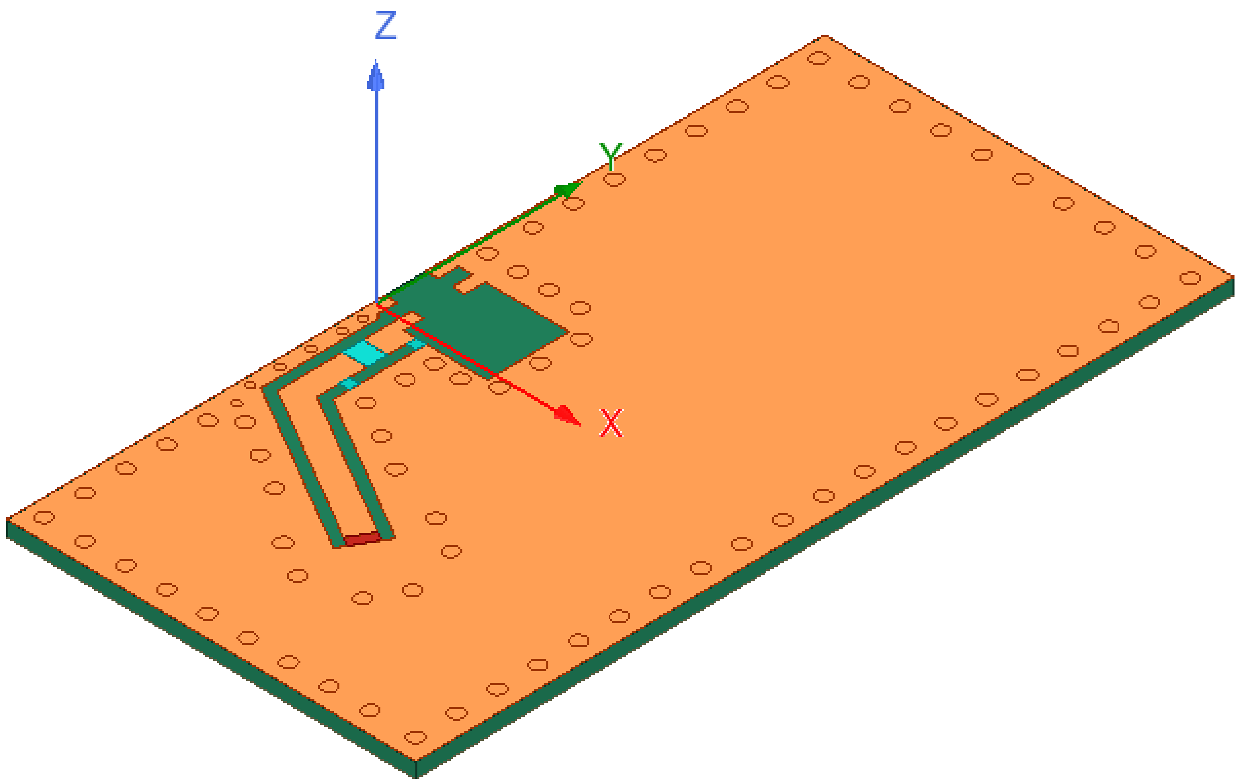
2. Right-click **Project*x*** at the top of the Project Manager and choose **Close** from the shortcut menu.

Open Project File

The project file for this exercise is located in the Help subfolder within the Ansys Electronics Desktop installation path. The *Program Files* folders have restricted access permissions, and besides, you shouldn't overwrite the sample models. Therefore, you will save the project to a suitable working folder after opening it.

1. On the **Desktop** ribbon tab, click **Open Examples**. Then:
   a. In the *Open* dialog box that appears, click the parent folder icon ( ) once to move up one level above the *Examples* folder.
   b. Double-click the **Help** folder and then the **HFSS** folder.
   c. Select the file **Chip_Antenna_Board.aedt** and click **Open**.

   The model appears in the Modeler window:



> **Note:**
>
> This project is deliberately incomplete. You need to add the chip antenna to complete the HFSS model. Additionally, you must insert a circuit design, add a port and lumped components, configure the Circuit design excitation, add a frequency sweep, tune the component values, push excitations from Circuit to HFSS, and generate reports. For these reasons, the project file is located in the *Help* folder instead of the *Examples* folder, which is reserved for completed projects.

Since you will not be constructing geometry for this exercise, the drawing grid and ruler are hidden in the project file.

2. On the **Desktop** ribbon tab, click **Save As**.
3. Navigate to a working folder of your choice. (Do not attempt to write to the program installation path.)

> **Note:**
>
> Optionally, you can click the **Create New Folder** icon ( ) within the *Save As* dialog box to create a new working folder in a suitable location.

4. Click **Save** to place a copy of the model in your working folder using the same file name.

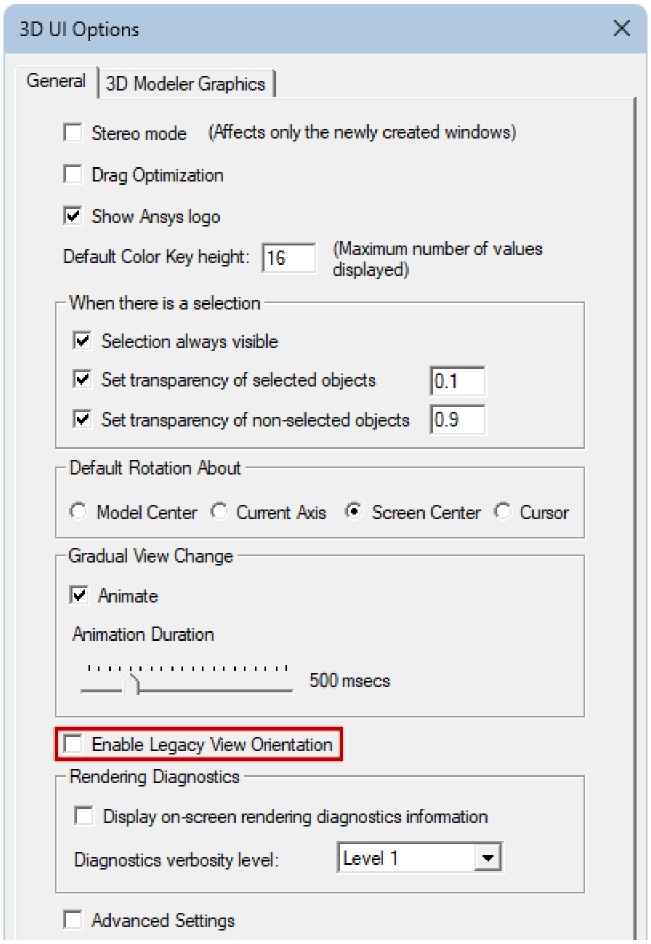
Verify 3D UI Options

Ensure that the legacy view orientation scheme is *not* being used, since the instructions and images in this guide are based on the new view orientation scheme introduced in release 2024 R1. To ensure that the model views you see match those shown in this guide, do the following:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:



3. Click **OK**.

Insert Chip Antenna

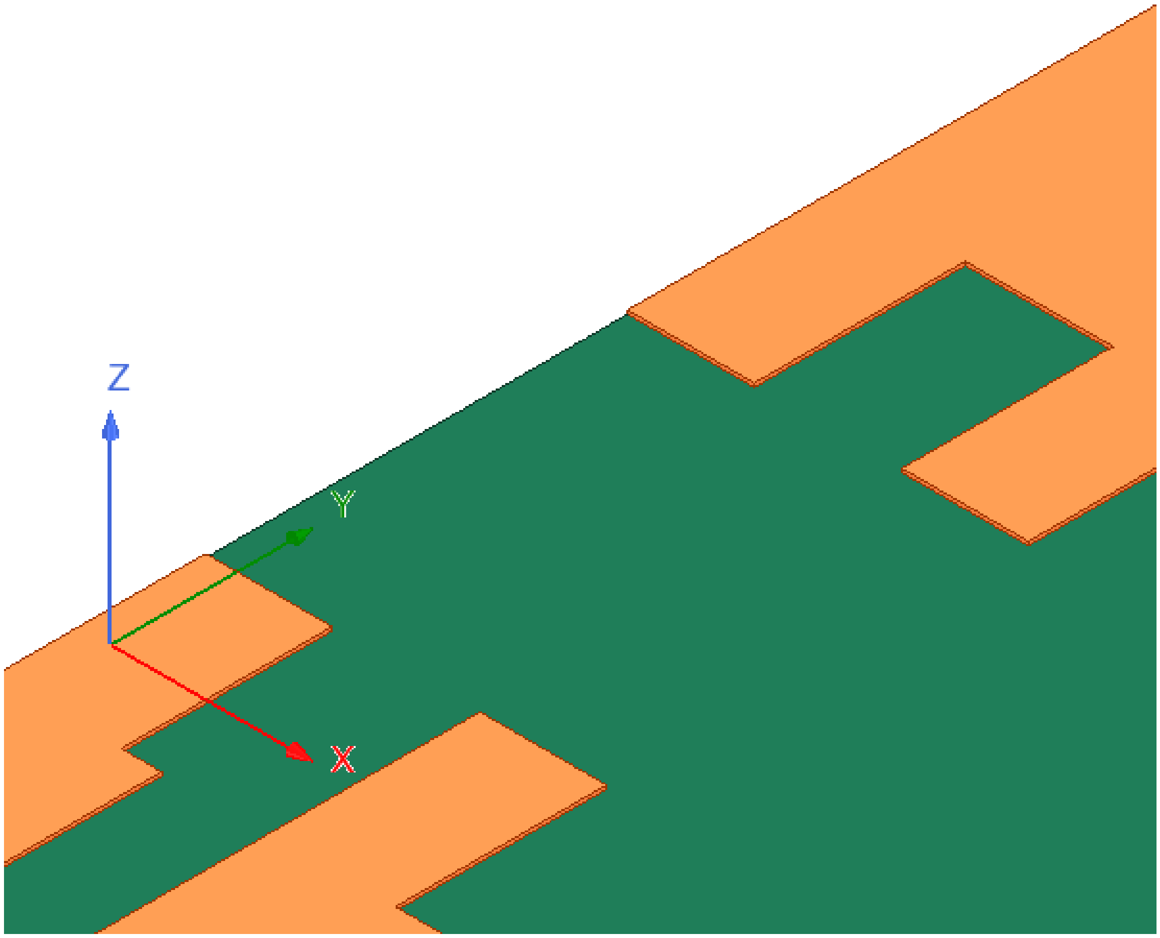
The chip antenna evaluation board has a relative coordinate system predefined (*AntCS*), which has its origin specified at the proper antenna insertion point. The global origin is located at the corner where the top (+Z), left (-Y), and back (-X) sides of the board meet. You will use the pre-defined AntCS coordinate system to add the antenna to the board.

> **Note:**
>
> The Johanson chip antenna is protected and encrypted. You cannot modify it in any way other than moving it's location within the model.

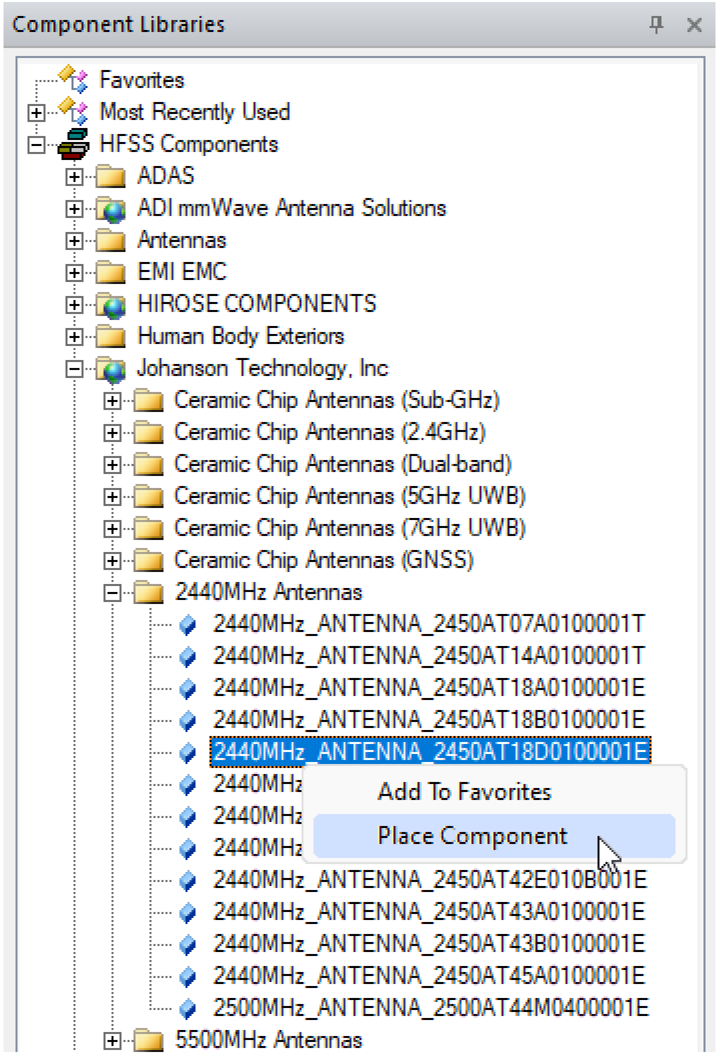
1. Under *Coordinate Systems* in the History Tree, ensure that **AntCS** is the working coordinate system. (A red "W" appears in the lower right corner of the icon when it is active.) If not, select **AntCS** to make it active.

2. With the cursor pointing at the AntCS origin, rotate the mouse wheel to zoom in closely to the antenna insertion area, as shown in the following image.

   The purpose is to ensure that you snap to the AntCS origin when placing the antenna and *not* to any nearby vertex or grid point.



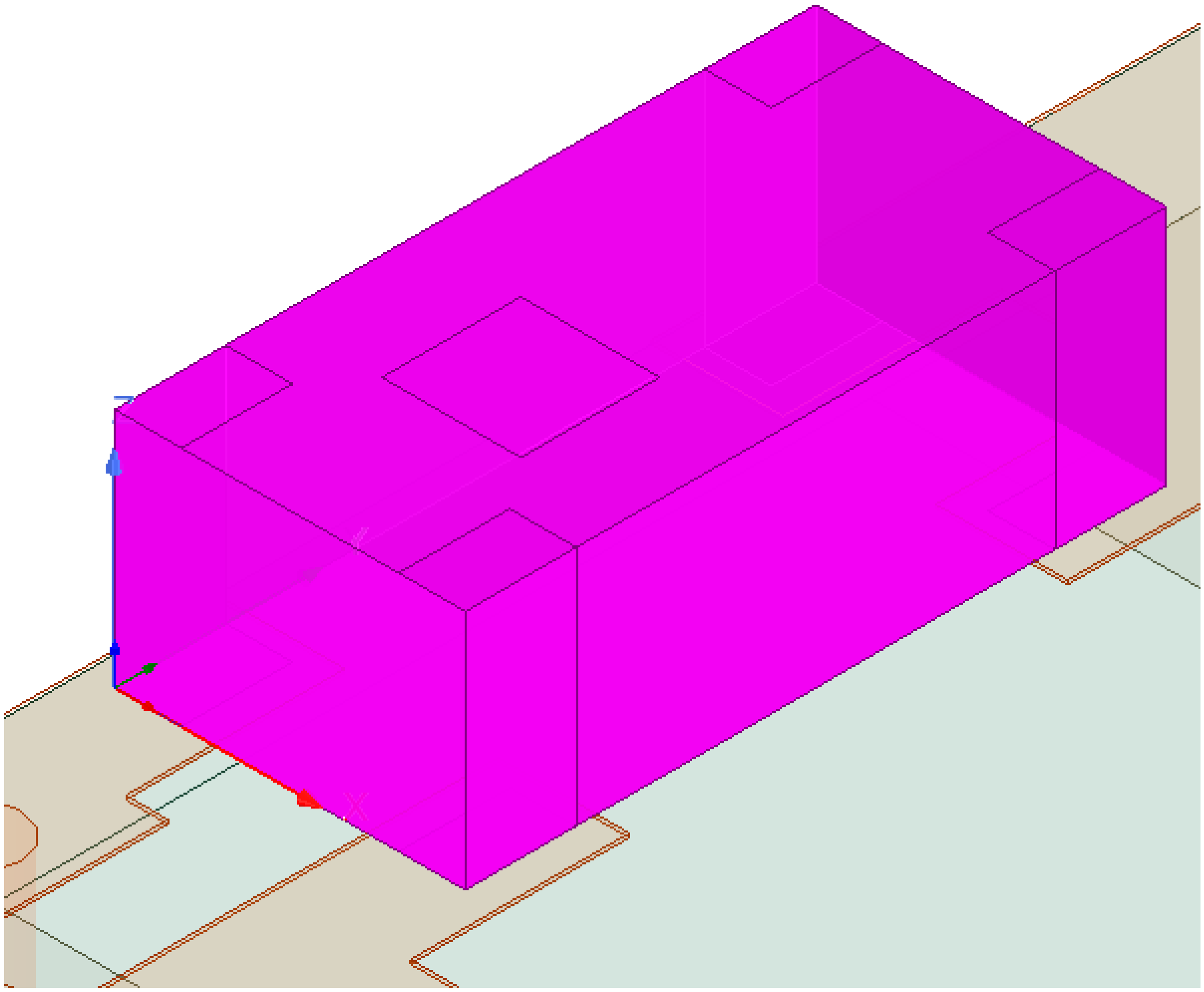
3. If the *Component Libraries* window is not already visible, use the menu bar to toggle its visibility. (Click **View** > **Component Libraries**.)
4. Expand the **HFSS Components** > **Johanson** > **2440MHz Antennas** folder in the *Component Libraries* window.
5. Right-click **2440MHz_ANTENNA_2450AT18D0100001E** and choose **Place Component** from the shortcut menu:



The *Insert 3D Component* dialog box appears.

6. Ensure that the **AntCS** is selected from the **Target Coordinate System** drop-down menu.
7. Click **OK**.
8. Point the cursor at the AntCS coordinate system origin and click the mouse to place the component insertion point there.

   The antenna appears at the correct position on the evaluation board:



> **Note:**
>
> Alternatively, you could click and drag the antenna from the library to the Modeler window, bypassing the *Insert CD Component* dialog box. Release the left mouse button and then click again to drop the component anywhere in the Modeler window.
>
> **Tip:**
>
> Additionally, instead of snapping to a grid point or any snapping point on the geometry, you can tab into the coordinate text boxes at the bottom of the user interface and type in the exact coordinates for placing the component. For example, to place the component at the origin of the selected coordinate system, press/-type: **Tab 0 Tab 0 Tab 0 Enter**, making sure that the coordinate entry mode is **Absolute**. Be careful not to bump the mouse while typing, or the coordinates will be reset to the cursor location.

9. Click in the Modeler window's background area to clear the selection.
10. On the **Draw** ribbon tab, click **Fit All**.
11. **Save** your project. (This command is available from all ribbon tabs and from the *File* menu.)

Set HPC Analysis Options

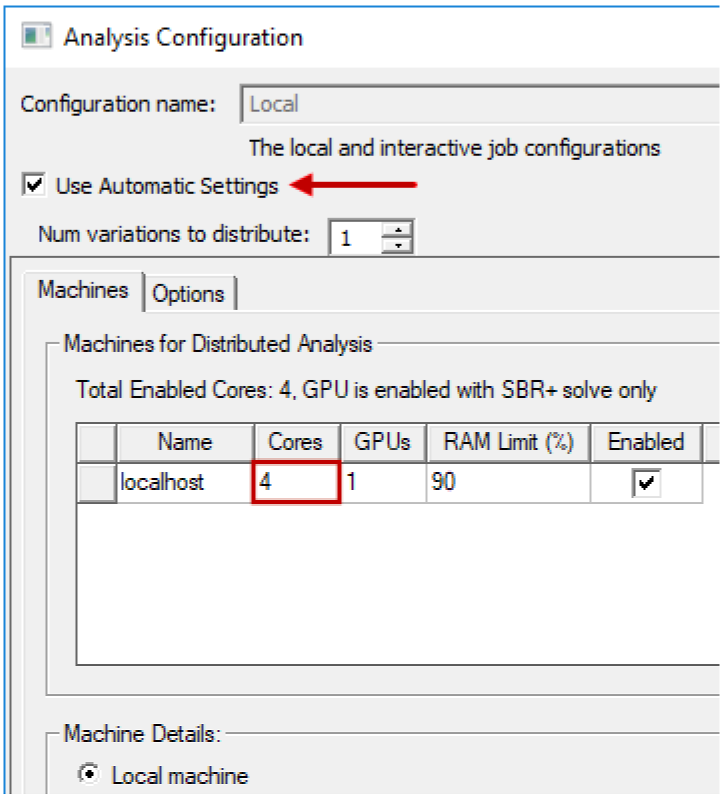
The high performance computing (HPC) capabilities of Ansys Electronics Desktop enable you to take advantage of multiple processing cores, which are common on all high-end computer workstations. Different portions of the analysis can thereby be solved in parallel, shortening the solution time.

> **Note:**
>
> The supported number of cores is determined based on hardware and licensing options. For the purpose of this exercise, use four cores, which is the supported number of cores for a basic software license.

1. On the **Simulation** ribbon tab, click **HPC Options**.

   The *HPC and Analysis Options* dialog box appears.

2. If the configuration named *Local* does not say **YES** in the **Active** column, select this configuration and click **Make Active**.
3. With **Local** still selected, click **Edit**.
4. In the Analysis Configuration dialog box that appears, specify the following settings:
    a. Select **Use Automatic Settings**
    b. In the **Cores** column for *localhost*, specify **4**.

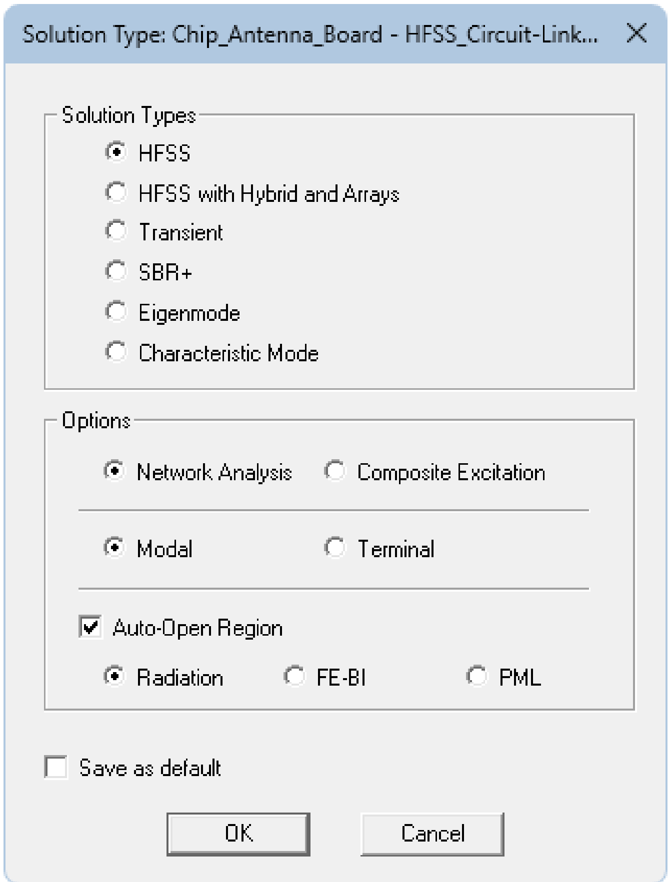


5. Click **OK** twice to close the *Analysis Configuration* and *HPC and Analysis Options* dialog boxes.
6.  **Save** your project.

Review the Simulation Setup

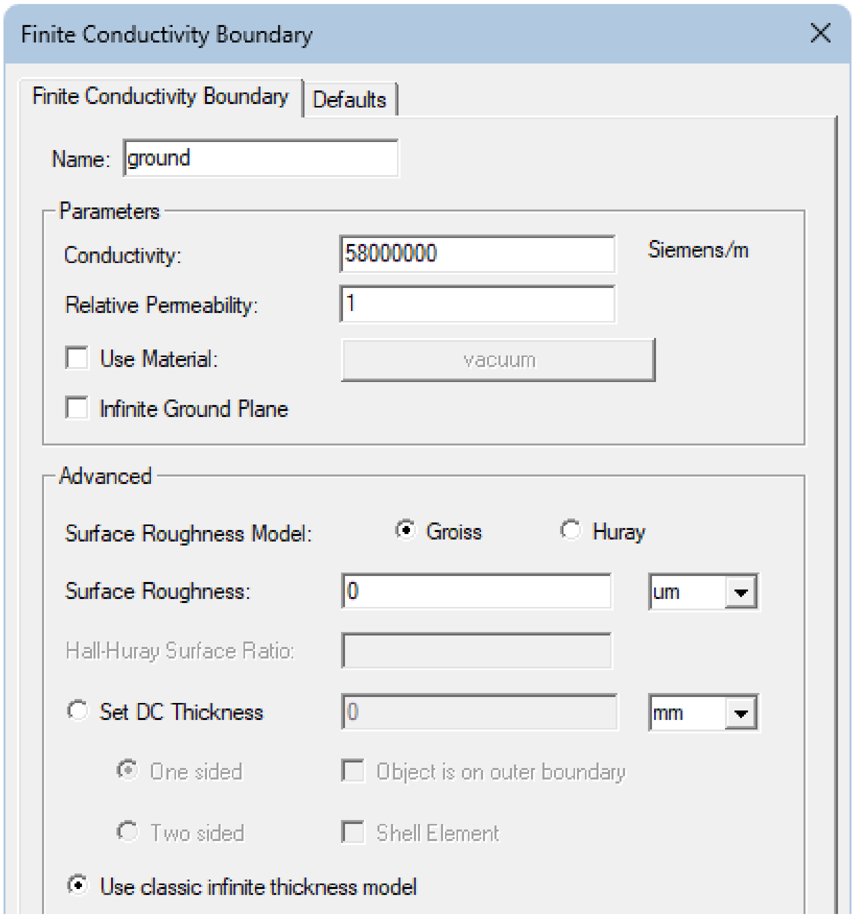
A boundary, four excitations, modified curved surface meshing, and the analysis setup and sweep are predefined in the source project. The input port and tuning components have all been defined as lumped ports, each assigned to a different sheet object. You can view the setup details as follows:

1. Right-click **HFSS_Circuit-Linked (Modal Network)** in the Project Manager and choose **Solution Type** from the shortcut menu. The settings are as shown in the following image:



Notice that this is an HFSS modal network solution and that it includes an automatic open region with a radiation boundary. You will not have to manually create an air or vacuum region for this design.

2. Click **Cancel** to close the *Solution Type* dialog box without making any changes.
3. Under *Boundaries* in the Project Manager, double-click **ground**.



A conductivity of 58,000,000 Siemens/m is assigned to the *ground* sheet object on the bottom face of the board, and **Use classic infinite thickness model** is selected.

4. Click **Cancel** to close the *Finite Conductivity Boundary* dialog box without making any changes.
5. Under *Excitations* in the Project Manager, select each of the four predefined excitations, one-at-a-time, to see their settings in the docked *Properties* window.

   All four excitations are **lumped ports** with **50 Ω full port impedance** and with **Deembed** and **Renormalize All Modes selected**. While each excitation is selected, you can also see the lumped port visualization on the model, including its defined integration line.

- **P1**:

Properties

Name	Value	Unit	Evaluated Value
Name	P1		
Type	Lumped Port		
Assignment	P1		
Impedance	50	ohm	50ohm
Num Modes	1		
Deembed	☑		
Renorm All Modes	☑		
Renorm Impedance Type	Impedance		
Renorm Imped	50	ohm	50ohm

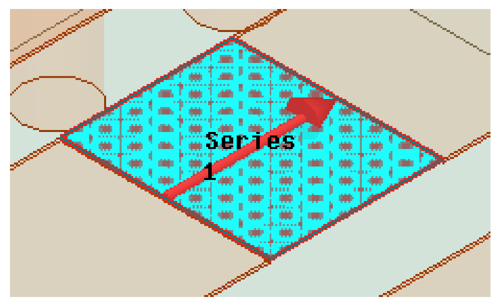
HFSS

- **Series**:

Properties

Name	Value	Unit	Evaluated Value
Name	Series		
Type	Lumped Port		
Assignment	series		
Impedance	50	ohm	50ohm
Num Modes	1		
Deembed	☑		
Renorm All Modes	☑		
Renorm Impedance Type	Impedance		
Renorm Imped	50	ohm	50ohm

HFSS

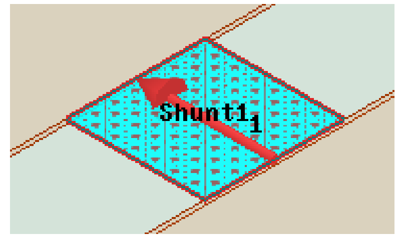


- **Shunt1**:

Properties

Name	Value	Unit	Evaluated Value
Name	Shunt1		
Type	Lumped Port		
Assignment	shunt1		
Impedance	50	ohm	50ohm
Num Modes	1		
Deembed	☑		
Renorm All Modes	☑		
Renorm Impedance Type	Impedance		
Renorm Imped	50	ohm	50ohm

HFSS

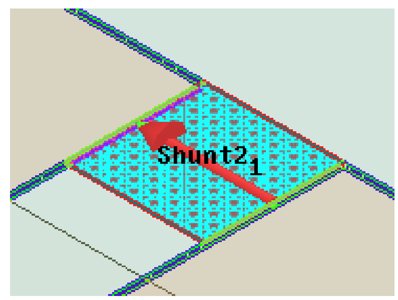


- **Shunt2**:

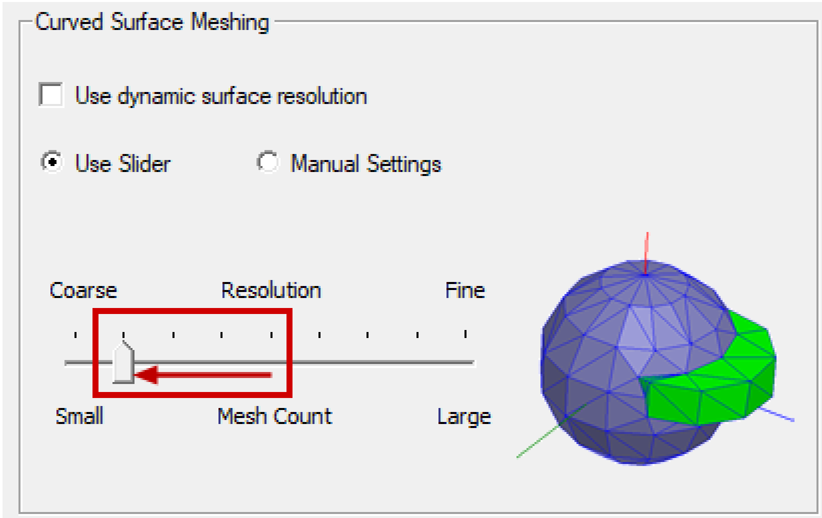
Properties

Name	Value	Unit	Evaluated Value
Name	Shunt2		
Type	Lumped Port		
Assignment	shunt2		
Impedance	50	ohm	50ohm
Num Modes	1		
Deembed	☑		
Renorm All Modes	☑		
Renorm Impedance Type	Impedance		
Renorm Imped	50	ohm	50ohm

HFSS

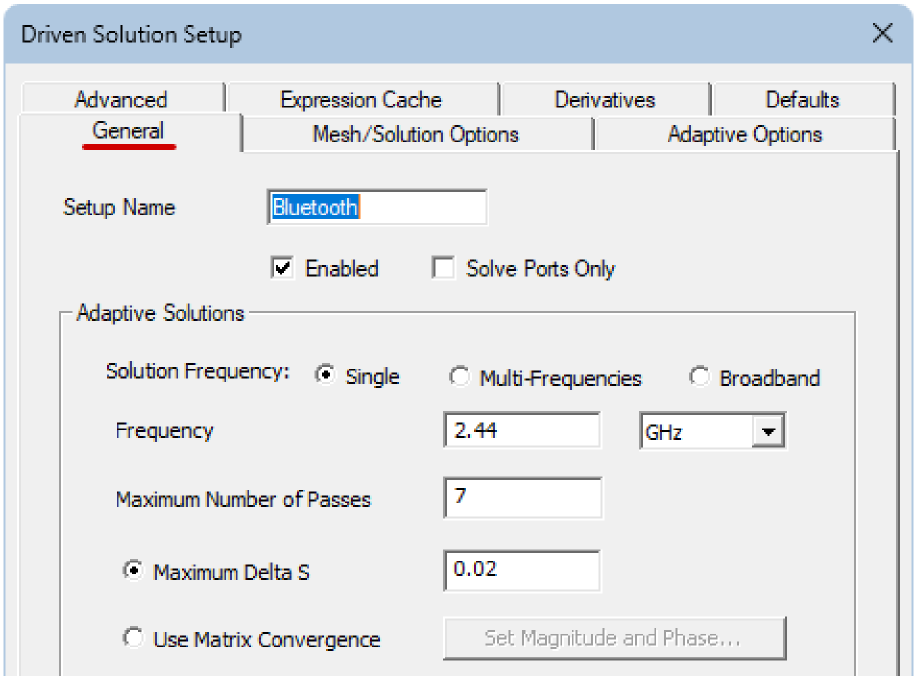


6. Right-click **Mesh** in the Project Manager and choose **Initial Mesh Settings**.



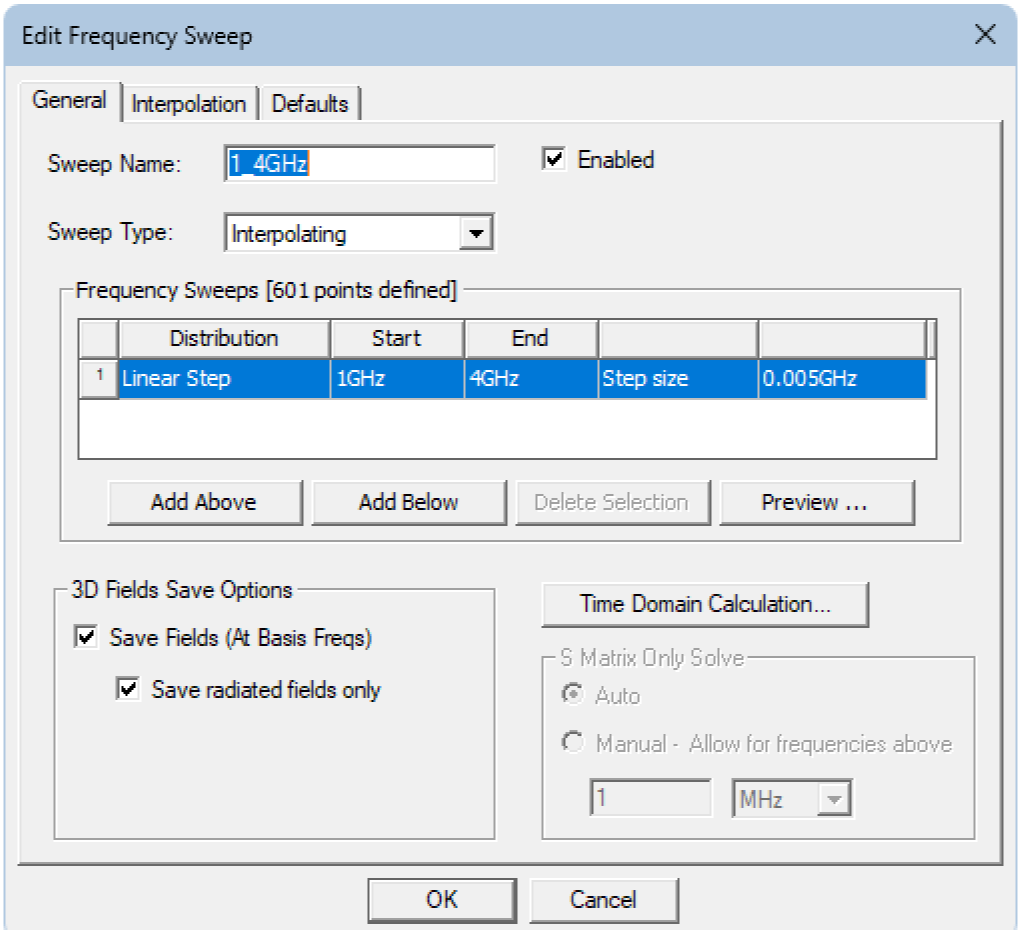
The *Curved Surface Meshing* slider has been moved three ticks coarser than the default position. This setting only affects curved geometry, which, for this example, is limited to the vias connecting the top and bottom ground planes. The default curved surface resolution would be unnecessarily fine for these vias and would significantly increase the element count, solution time, and memory requirement without improving the results.

7. Under *Analysis* in the project manager, double-click **Bluetooth** to see the analysis properties, which are summarized as follows:
   - **General** tab:
     - Adaptive Solution Frequency = 2.44 GHz
     - Maximum Number of Passes - 7
     - Maximum Delta S = 0.02



- **Advanced** tab: **Save Fields** is enabled for all objects:
- **Mesh/Solution Options** tab: **Auto-select Direct/Iterative** is selected. All other settings in this tab are the defaults.

8. Click **Cancel** to close the *Driven Solution Setup* dialog box without making any changes.
9. Under *Analysis* > *Bluetooth* in the Project Manager, double-click **1_4GHz** to see the frequency sweep definition.



Here you can see that the sweep is in linear steps of 0.005 GHz from 1 to 4 GHz, with a total of 601 points, and the radiated fields will be saved.

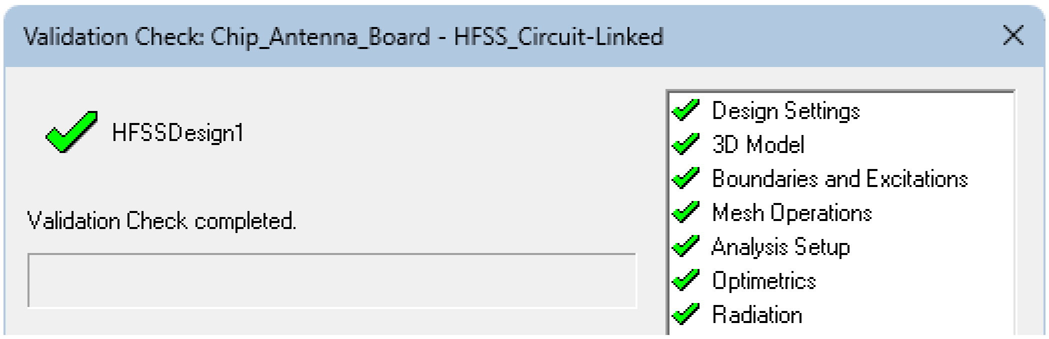
Additionally, three project variables have been predefined. We will look at those in a later procedure.

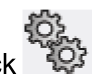
3 - Validate and Analyze HFSS Design

Verify that there are no model setup errors and run the HFSS simulation as follows:

1. On the **Simulation** ribbon tab, click  **Validate**.

   The *Validation Check* window appears, and it should show no warnings or errors:

2. Click **Close** to dismiss the *Validation Check* window.
3. On the **Simulation** ribbon tab, click  **Analyze All**.

   > **Note:**
   >
   > The *Validate* and *Analyze All* commands are also available from the *HFSS* menu and from the shortcut menu that appears when you right-click the HFSS design in the Project Manager.

4. If the *Message Manager* and *Progress* windows are not displayed, click **Show Messages** and **Show Progress**, at the right end of the status bar, to see the solution messages and progress.
5. Optionally, click and drag the borders to resize these windows as preferred.

The HFSS solution takes about four minutes or less to solve using four cores on a reasonably current computer workstation.

4 - Set Up and Analyze Circuit Design

In this chapter, you will perform the following tasks:

- Insert a Circuit design into the project
- Add a link to the HFSS design in the Circuit design
- Add a port to the HFSS model's schematic in the Circuit design
- Add lumped components to the Circuit design
- Connect components in the schematic
- Configure the Circuit design excitation
- Review the port source properties

Insert Circuit Design

You will add the matching network components and perform their tuning in a Circuit design. Add one to the project as follows:

1. When the HFSS analysis has finished solving, **Save** the project.
2. On the **Desktop** ribbon tab, click **Circuit**.

   The *Workflow* dialog box appears.
3. In the list at the right side of the *Workflow* dialog box, ensure that **None** is selected and click **OK**.

   An empty *Schematic* window appears and a *Circuit1* branch is added to the tree in the Project Manager.
4. In the Project Manager, collapse the **HFSS_Circuit-Linked (Modal Network)** branch, and expand the **Circuit1** branch (if it is not already expanded).
5. Right-click **Circuit1** and choose **Rename**.
6. Type **Matching Network** and press **Enter**.

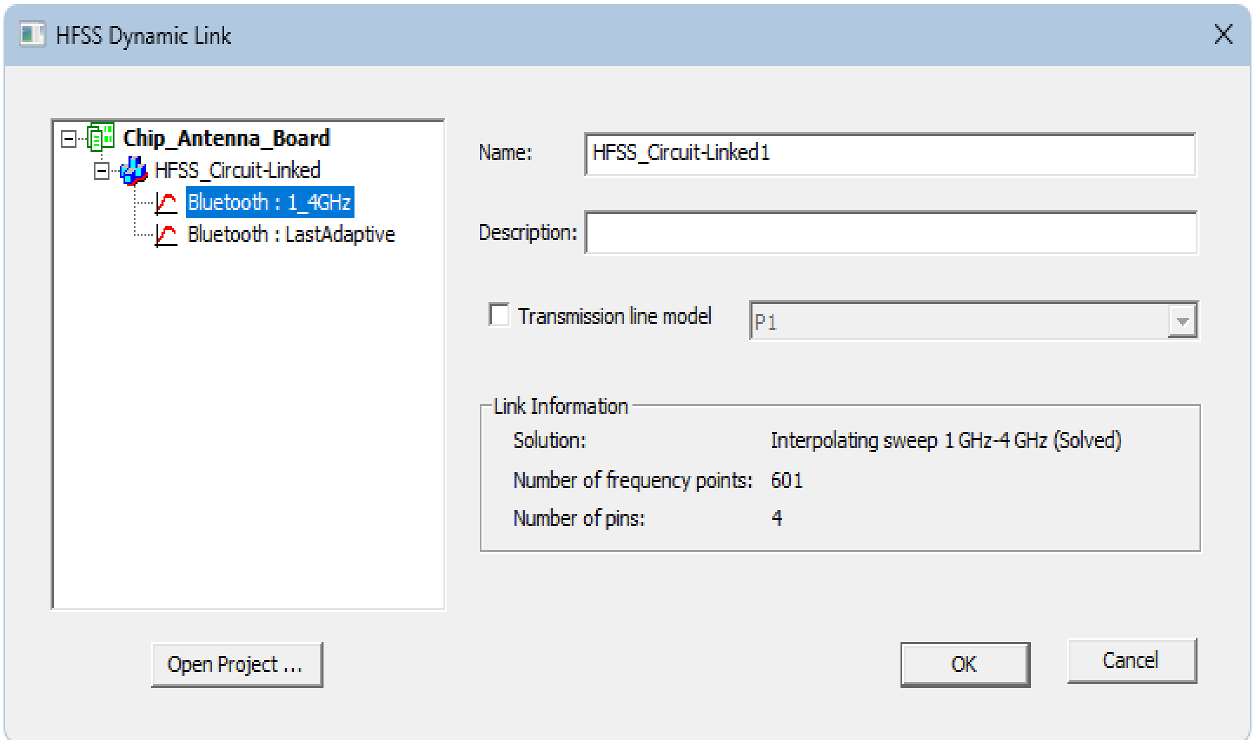
Add HFSS Model to Circuit

In this procedure, you will copy the HFSS design of the evaluation board and chip antenna into the Circuit design.

1. In the Project Manager, right-click **Matching Network** and choose **Add Subcircuit** > **Add HFSS Link** from the shortcut menu.

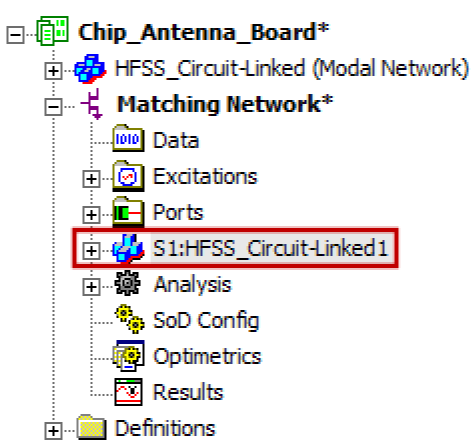
   The *HFSS Dynamic Link* dialog box appears.

2. On the left side of the dialog box, under *Chip_Antenna_Board* > *HFSS_Circuit-Linked*, select the frequency sweep (**Bluetooth : 1_4GHz**):



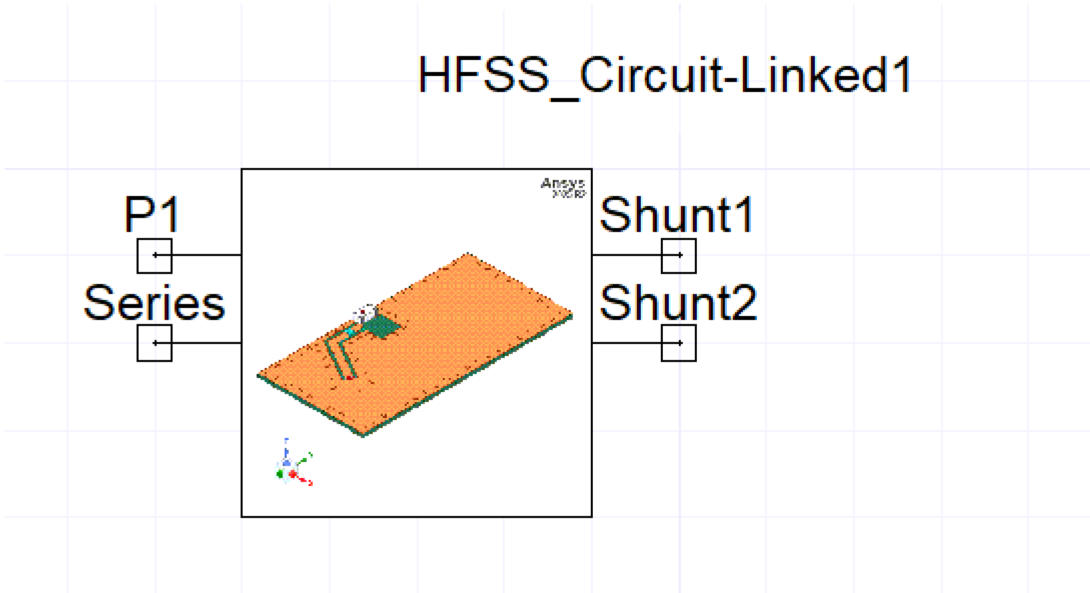
3. Click **OK**.

   A subcircuit model of the HFSS design is inserted into the *Matching Network* (Circuit) design in the Program Manager. A number "1" has been appended to the end of the source HFSS design name.



The HFSS model also appears in the *Schematic* window.

4. On the **Schematic** ribbon tab, click **Fit All**. Then, click **Zoom Out** twice to make room in the *Schematic* window for adding some components.

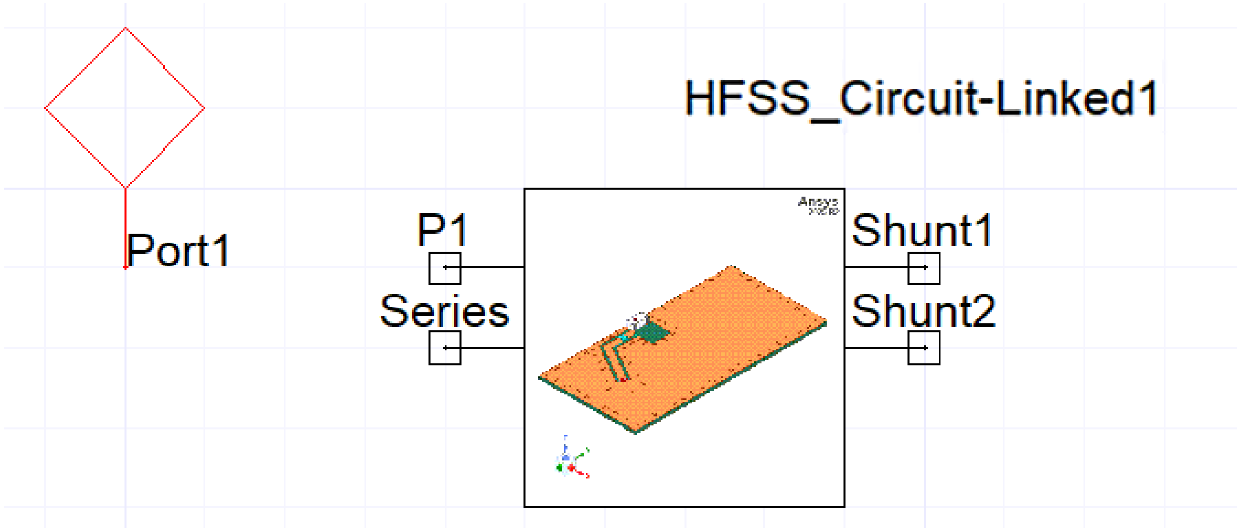


> **Note:**
>
> Zoom in or out and scroll in the Schematic window as needed while completing the circuit. The mouse buttons and scroll wheel work somewhat differently in the *Schematic* window than they do in the 3D Modeler. Roll the mouse wheel to zoom in or out, as in the 3D Modeler. Press **Shift** while rolling the wheel, or use the scroll bar on the right edge of the window, to scroll the schematic vertically. Use the bottom scroll bar to scroll horizontally.

Add Port to Schematic

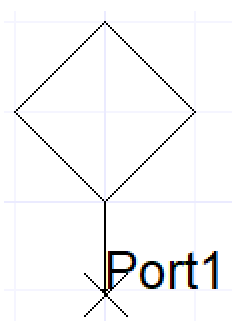
Add an interface port to the schematic. Then, connect it to pin 1 of the HFSS evaluation board model you added to the schematic in the last procedure.

1. On the **Schematic** ribbon tab, click **Interface port**.
2. Place the port symbol on the schematic in the approximate position shown below. When you click, the insertion point of the symbol will snap to the nearest grid point.



3. Right-click and choose **Finish** to exit the *Interface port* command (alternatively, press **Space** or **Esc**).
4. Point to the bottom end of the **Port1 pin** (but do not click yet).

   The cursor changes to an X when near a valid connection point.

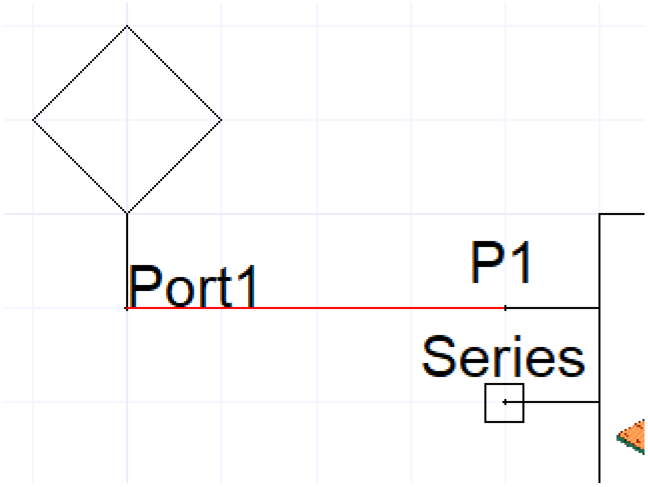


5. Click this point.

   The **Wire** command is started automatically, and the start point of the wire snaps to the *Port1* pin.

6. In the same manner, click the mouse at the pin labeled **1**. (A small circle indicates the snapping point.)

   The completed connection wire should match the following image:

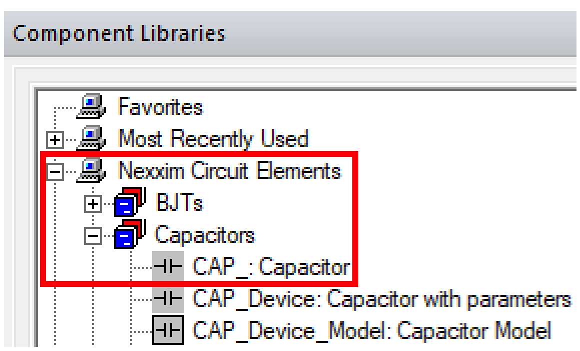


Because the line completes a connection to a pin, the segment is terminated. Otherwise, if you clicked at a non-connection point (say to change directions) you can continue to draw line segments until a valid termination point is reached.

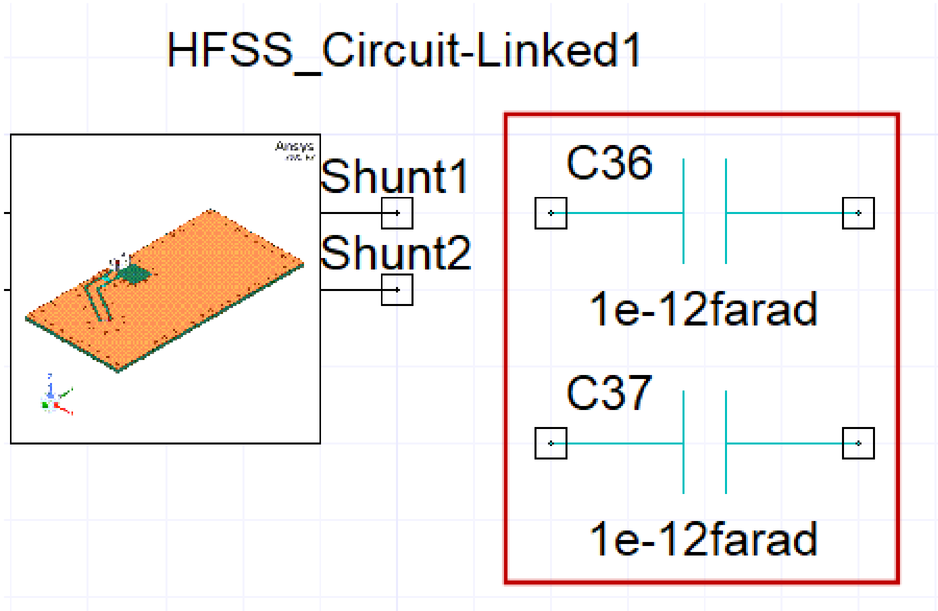
Add Components to Schematic

Next, you will add two capacitors and one inductor to the schematic. You will also specify the appropriate parameters and values.

1. Navigate to the **Components** tab of the *Component Libraries* window.
2. Under *Nexxim Circuit Elements*, expand the **Capacitors** branch.



3. Click and drag the component labeled **CAP_: Capacitor** onto the right side of the schematic. Then move the cursor and click again to place a second one on the right side, below the first. Locate the two capacitors at the approximate positions shown below:



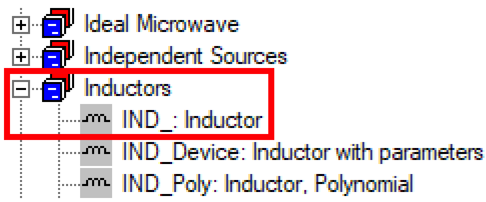
4. Right-click and choose **Finish** (or press **Space** or **Esc**) to terminate the component placement operation.
5. Click the upper capacitor to select it.

   The settings associated with this component appear in the docked *Properties* window.

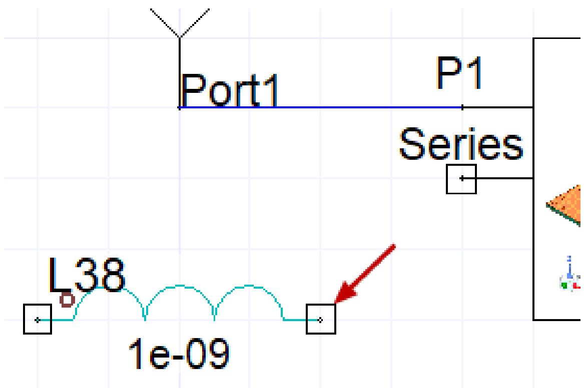
6. In the **Param Values** tab of the docked *Properties* window, replace the default numerical *Value* of **C** with a variable, **C1**, and press **Enter**.
7. In the *Add Variable* dialog box that appears, do the following:
   a. Select **Capacitance** from the **Unit Type** drop-down menu (if it is not already selected).
   b. Select **pF** from the **Units** drop-down menu.
   c. Type **2** in the **Value** text box.
   d. Click **OK**.
8. In the **Component** tab of the docked *Properties* window, change the **InstanceName** *Value* to **C1** if it is not already specified as such.

   Setting both the InstanceName and the variable name to *C1* makes it easy to tell which variable goes with which component. You'll do the same for the remaining components.

9. Repeat steps 5 through 8 for the lower capacitor. This time, specify **C2** for both the variable and the InstanceName and set the value of variable **C2** to 1.5 pF.
10. Under *Nexxim Circuit Elements*, expand the **Inductors** branch.

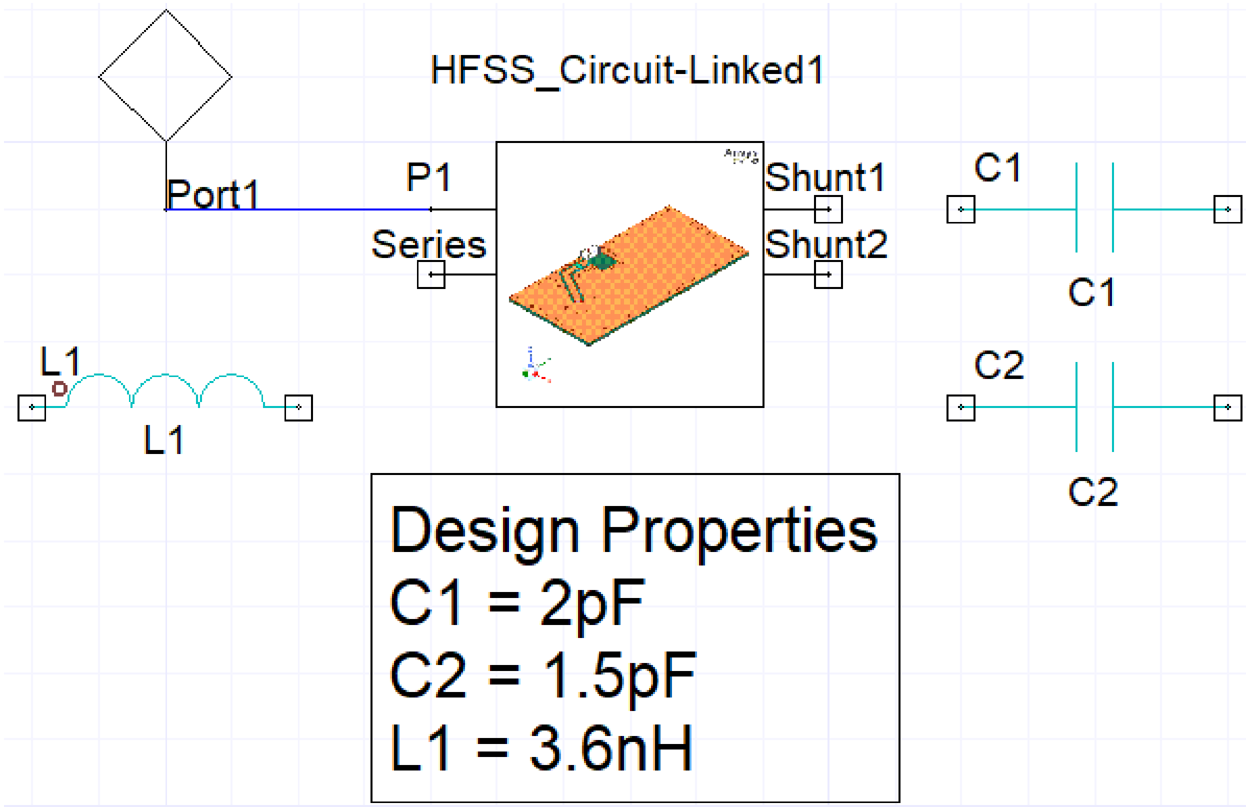


11. Double-click the component labeled **IND_: Inductor**.
12. Right-click at the desired inductor insertion point on the left side of the schematic (as shown below) and choose **Place and Finish** from the shortcut menu.



13. Click the inductor you just placed on the schematic to select it.
14. In the **Param Values** tab of the docked *Properties* window, replace the default *numerical value* of **L** with a variable, **L1**, and press **Enter**.
15. In the *Add Variable* dialog box that appears, do the following:
    a. Select **Inductance** from the **Unit Type** drop-down menu (if it is not already selected).
    b. Select **nH** from the **Units** drop-down menu.
    c. Type **3.6** in the **Value** text box.
    d. Click **OK**.
16. In the **Component** tab of the docked *Properties* window, change the **InstanceName** *Value* to **L1** if it is not already specified as such.
17. In the Schematic window, click and drag the **Design Properties** box and the **HFSS_Circuit-Linked1** name to approximately center them at the top and bottom of the schematic, respectively.
18. Click in the background to clear the current selection.

    The schematic should resemble the following image:

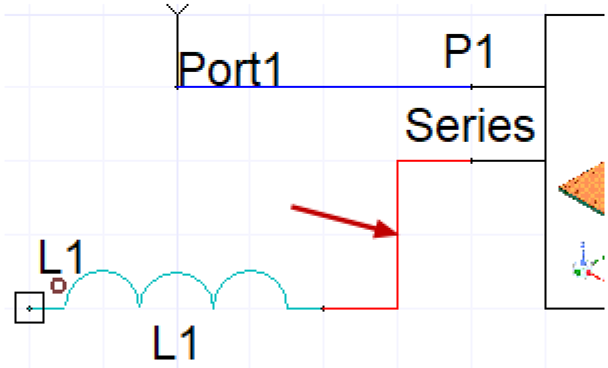


Connect Components in Schematic

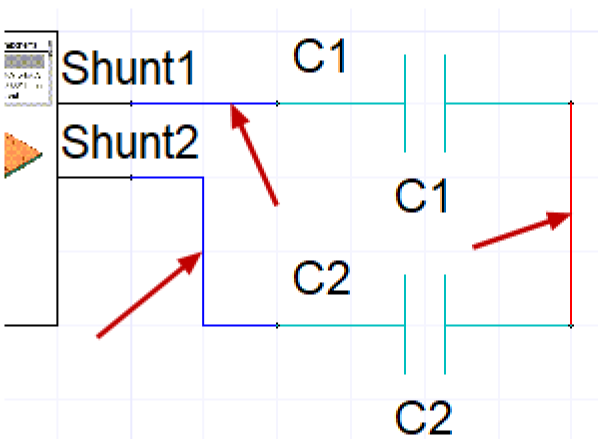
Next, you will connect the discrete components just added to the schematic to the appropriate points on the evaluation board.

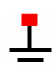
1. Click the **right** end of inductor **L1** to start a connection line, click two intermediate points, and click pin **Series** to terminate the connection.

> **Note:**
>
> In a proper schematic, diagonal connection lines are avoided. The intermediate points enable a multi-segment connection in which all line segments are orthogonal. This connection should resemble the following image when completed:

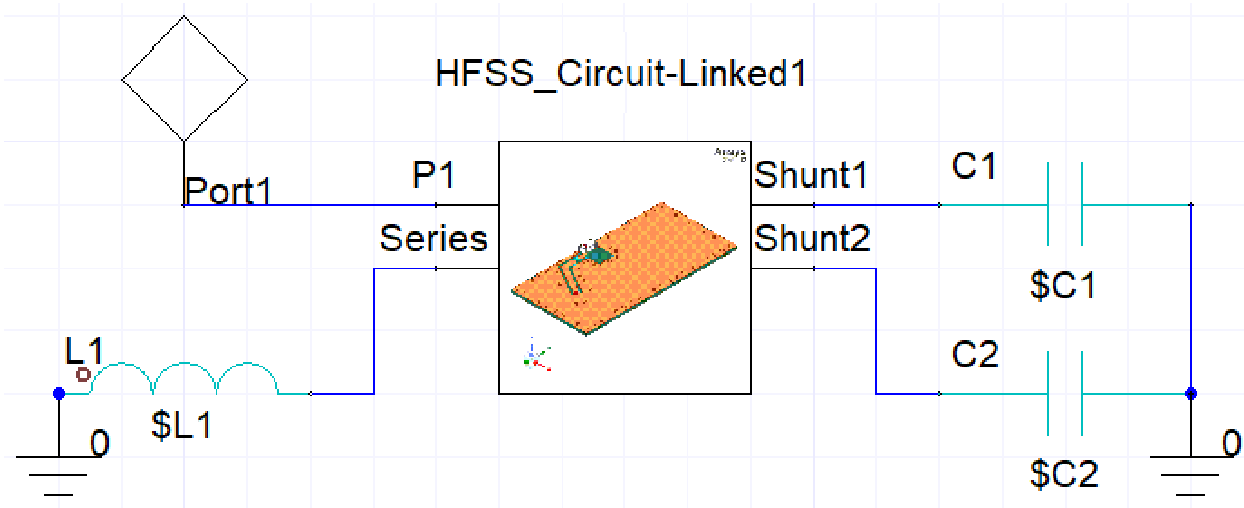


2. In the same manner, add the following three connections:
    - **Left** end of **C1** to pin **Shunt1**
    - **Left** end of **C2** to pin **Shunt2**
    - **Right** end of **C1** to **Right** end of **C2**



3. On the **Schematic** ribbon tab, click  **Ground**.
4. Click to connect the Ground to the **left** end of **L1**.
5. Click to connect a second Ground to the **right** end of **C2**.
6. Press **Space** to exit the *Ground* command.
7. Click in the background to clear the selection.

    The finished schematic should resemble the following image:



8.  **Save** the project.

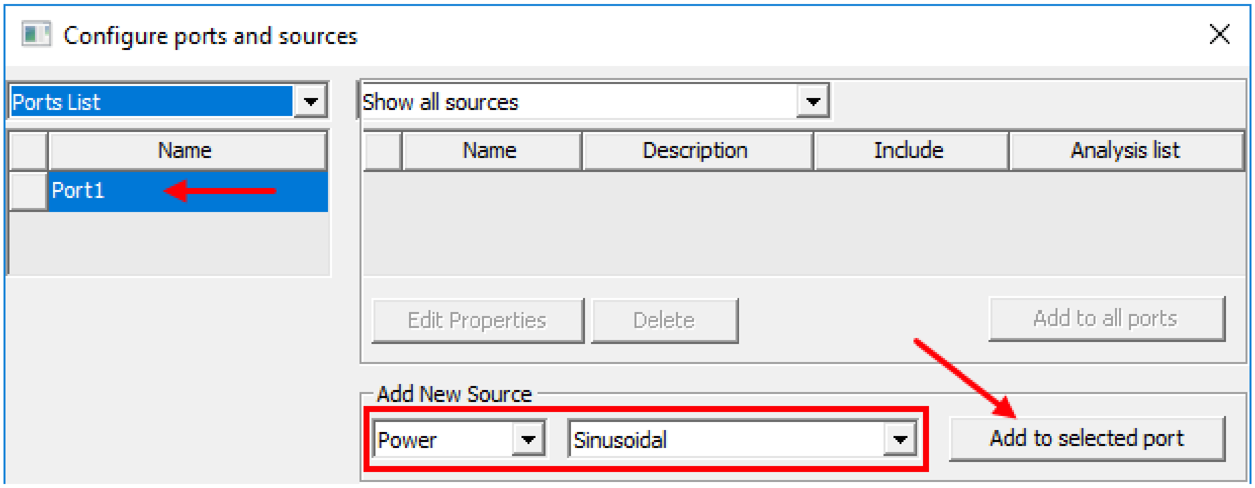
Configure Circuit Excitation

In this procedure, you will add a 1 volt peak magnitude AC power source to the circuit's *Port1* configuration.

1. Under *Matching_Network* in the Project Manager, right-click **Excitations** and choose **Configure Sources** from the shortcut menu.

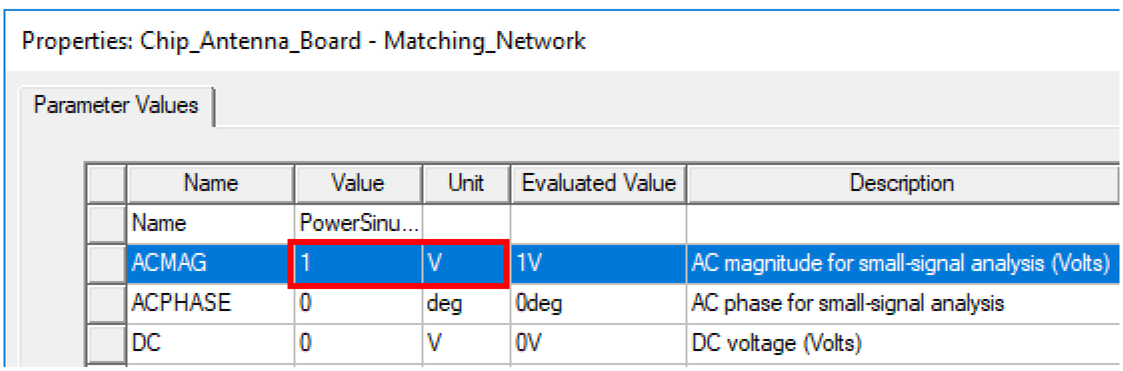
   The *Configure ports and sources* dialog box appears. Port1 is already selected because it's the only item in the Ports List.

2. From the drop-down menus in the *Add New Source* section of the dialog box, choose **Power** and **Sinusoidal** (if not already selected). Then click **Add to selected port**:



The *Properties* dialog box appears.

3. For the parameter **ACMAG**, specify a value of **1 V**:

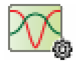


> **Note:**
>
> This voltage is the peak magnitude of the AC signal, not peak-to-peak. The peak-to-peak amplitude = 2 * ACMAG, and the average amplitude = ACMAG / √2.

4. Click **OK** to close the *Properties* dialog box.
5. Click **OK** to close the *Configure ports and sources* dialog box.

Add Frequency Sweep and Analyze

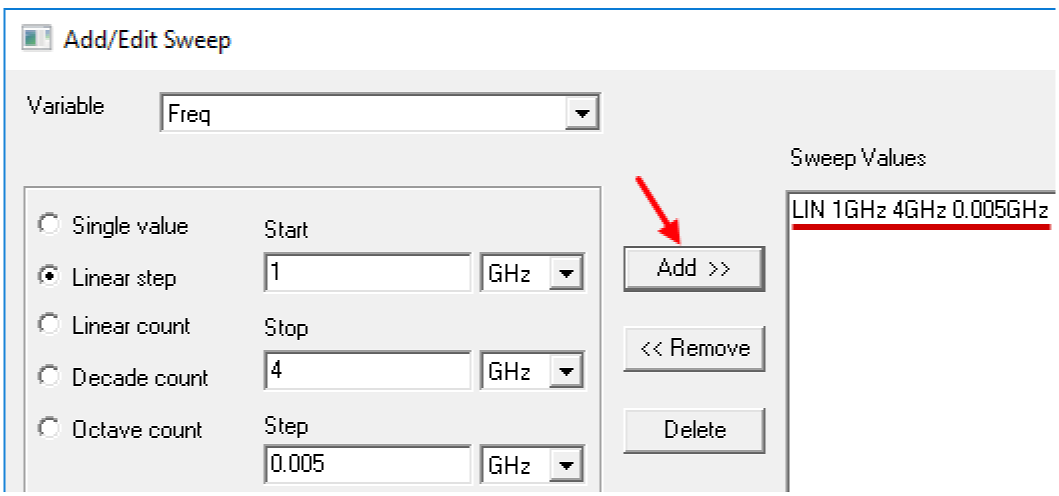
You will now set up a frequency sweep in the Circuit design spanning from 1 to 4 GHz in 0.005 GHz increments.

1. On the **Simulation** ribbon tab, click  **LNA**. (If *LNA* is not shown as the default action for any of the simulation type drop-down menus, select it from the first drop-down menu.)

   After clicking the icon, the default action is to set up a *Nexxim, Linear Network Analysis (LNA)*.

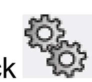
   The *Linear Network Analysis, Frequency Domain* dialog box appears.

2. Immediately below the *Sweep Variables* table, click **Add**.
3. In the *Add/Edit Sweep* dialog box that appears, specify the following settings:
   a. Select the **Linear step** option.
   b. **Start** = **1 GHz**
   c. **Stop** = **4 GHz**
   d. **Step** = **0.005 GHz**
4. Click **Add** to populate the *Sweep Values* box.



5. Click **OK** to close the *Add/Edit Sweep* dialog box.
6. Click **OK** to close the *Linear Network Analysis, Frequency Domain* dialog box.

   The Circuit design is ready to be analyzed.

7. On the **Simulation** ribbon tab, click  **Analyze**.

   The Circuit simulation only takes a second or two to finish.

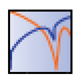
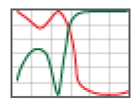
5 - Tune Matching Network

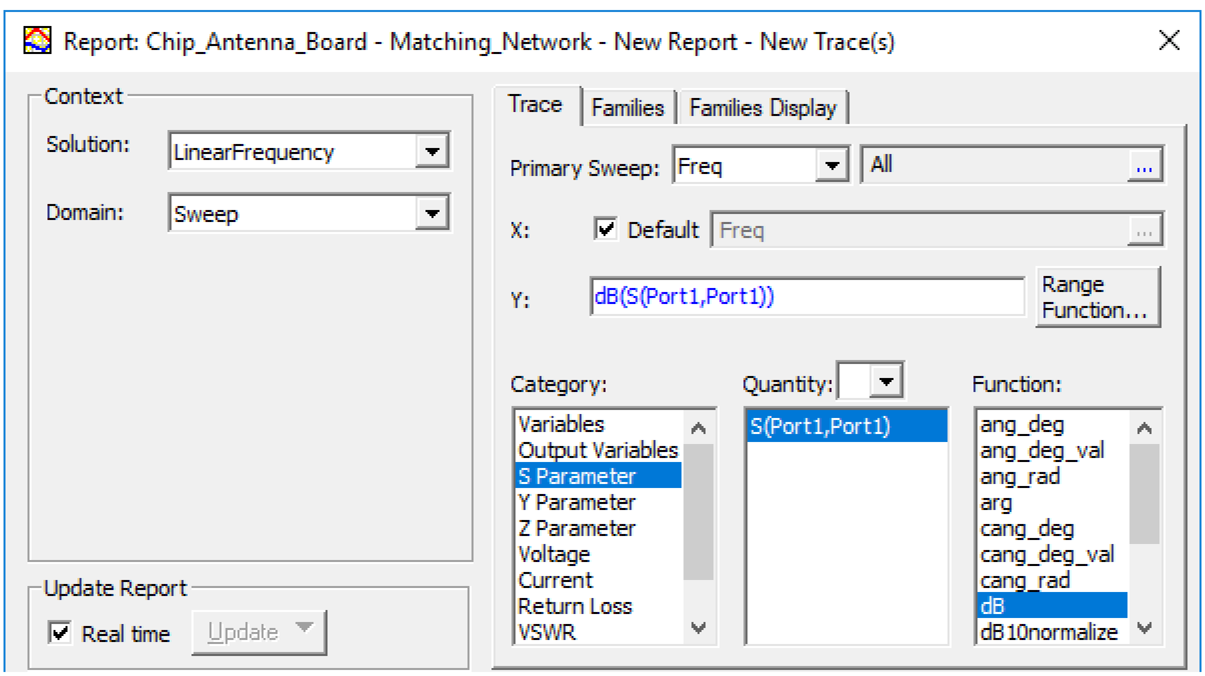
In this chapter, you will tune the component values in the matching network to minimize the signal reflection at 2.44 GHz. The goal is for the circuit and antenna to be resonant at this frequency. The process involves the following steps:

- Create an S-parameter plot
- Prepare the S-parameter plot for circuit tuning
- Select the variables to be tuned
- Tune component values

Create S-Parameter Report

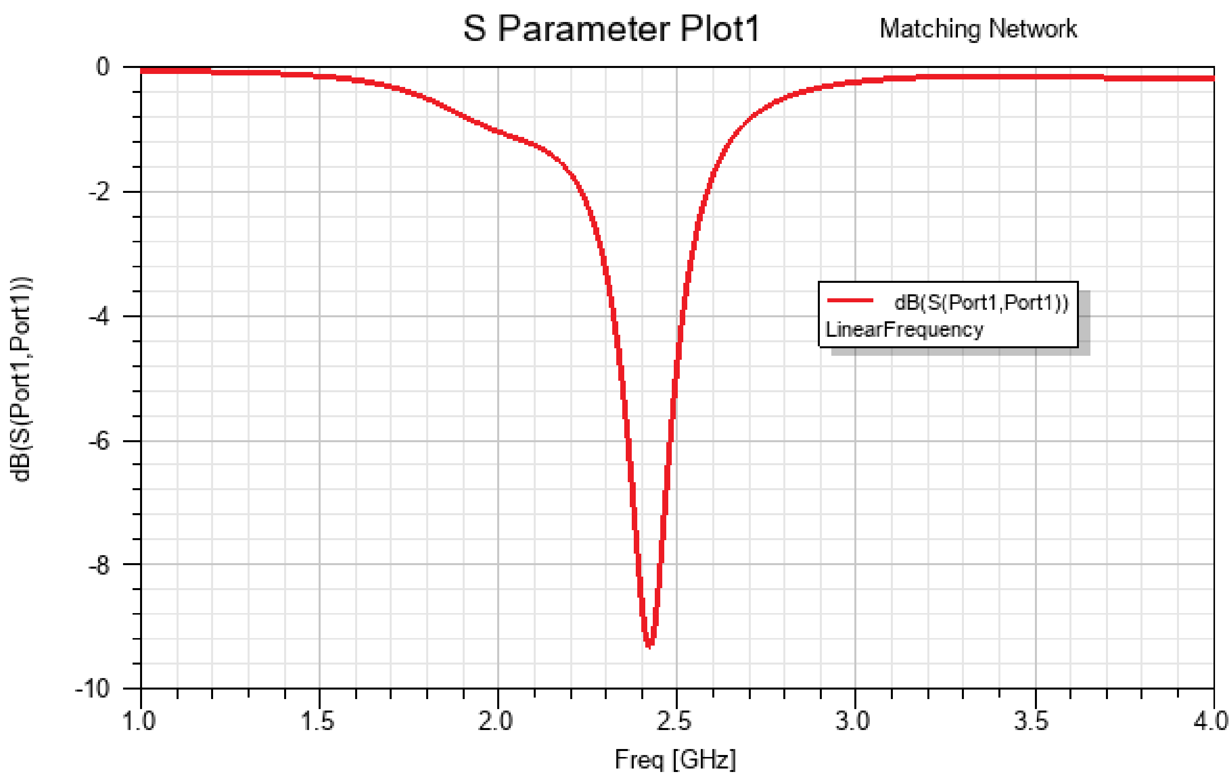
The first step in tuning the circuit is to create an S-parameter report of the signal reflection (or return loss) at Port1. This plot will be used as the basis of the component tunings to be performed. The signal reflection will be minimal at the resonant frequency of the circuit. Varying the capacitance and inductance values will alter the resonant frequency of the circuit.

1. On the **Results** ribbon tab, click  **Standard Report** >  **2D**.
2. In the *Report* dialog box that appears, ensure that all settings are as shown in the following image:



3. Click **New Report** and click **Close**.

   S Parameter Plot 1 appears in a new window:



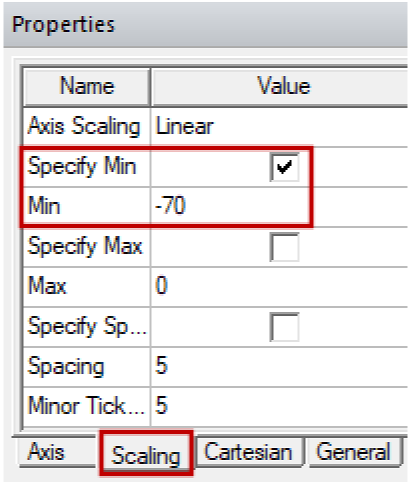
Prepare Report for Tuning

In this procedure, you'll modify the Y scaling of the S-parameter plot and add an X-marker at the desired resonant frequency of 2.44 GHz.

1. Click on the **Y axis** of *S Parameter Plot 1* to select it.

   The Y axis becomes bold, and the associated settings appear in the docked *Properties* window.

2. In the **Scaling** tab of the docked *Properties* window, make the following changes:
   a. Select the **Specify Min** option.
   b. Change the **Min** value to **-70** and press **Enter**.



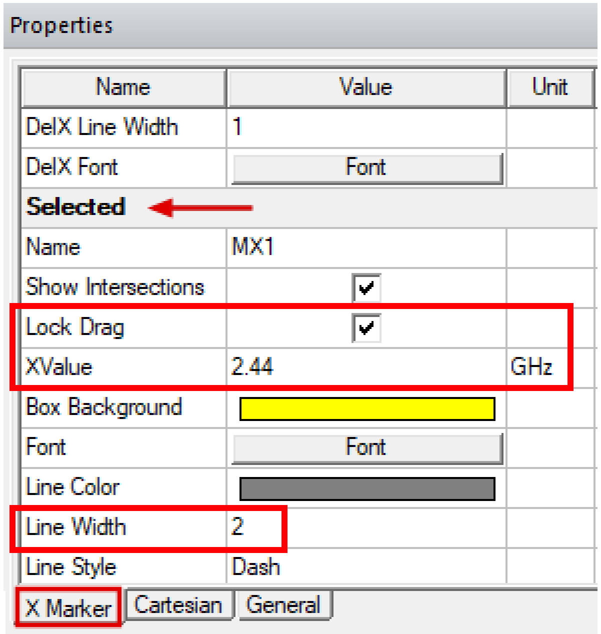
3. Right-click in the *S Parameter Plot 1* window and choose **Marker** > **Add X Marker**.

   A vertical marker line is added to the plot. The bottom yellow box indicates the frequency value and the top one the corresponding S-parameter value.

4. Click inside the **bottom** yellow marker box (*frequency*).

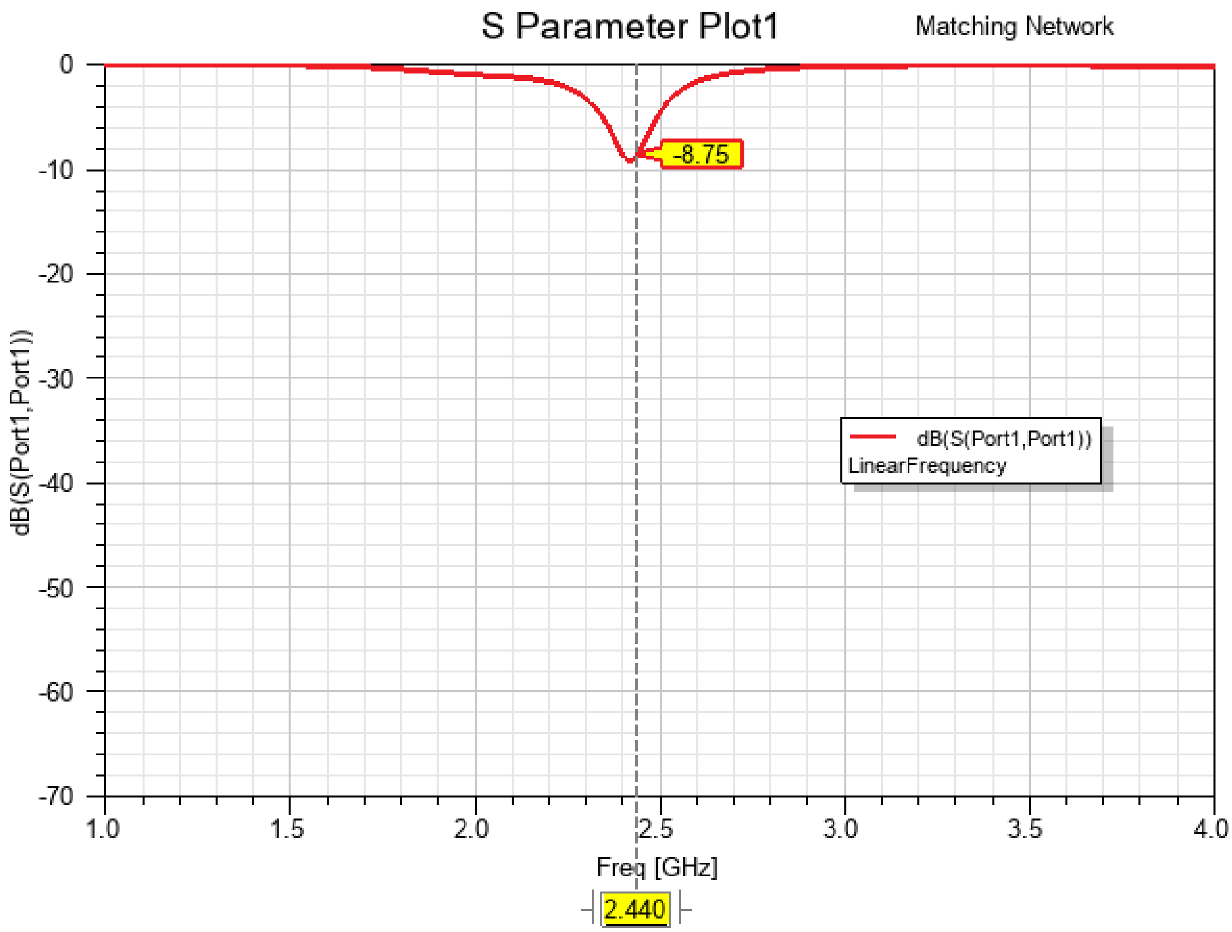
   The X marker settings appear in the docked *Properties* window.

5. In the **X Marker** tab of the docked *Properties* window, scroll down to the properties under the *Selected* heading (if necessary) and make the following changes:
   a. Select the **Lock Drag** option.
   b. Change **XValue** to **2.44 GHz**.
   c. Change **Line Width** to **2** and press **Enter**.



6. Click in the plot background area to clear the selection.

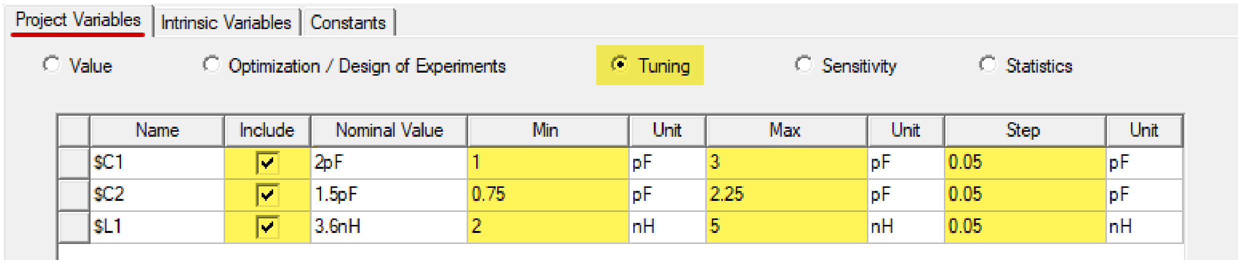
   The modified plot should look like the following image:



Select Variables for Tuning

You must specify which variables you want to include in the tuning process. You will include all three of the previously defined variables: *C1, C2,* and *L1*.

1. In the Project Manager, right-click **Matching Network** and choose **Design Properties** from the shortcut menu.
2. In the **Local Variables** tab of the *Properties* dialog box that appears, make the following changes:
   a. Select the **Tuning** option.
   b. Select the checkboxes in the **Include** column for all three variables (**Shunt1, Shunt2,** and **Series**).
   c. In the **Min**, **Max**, and **Step** columns, set the values as shown in the following image:



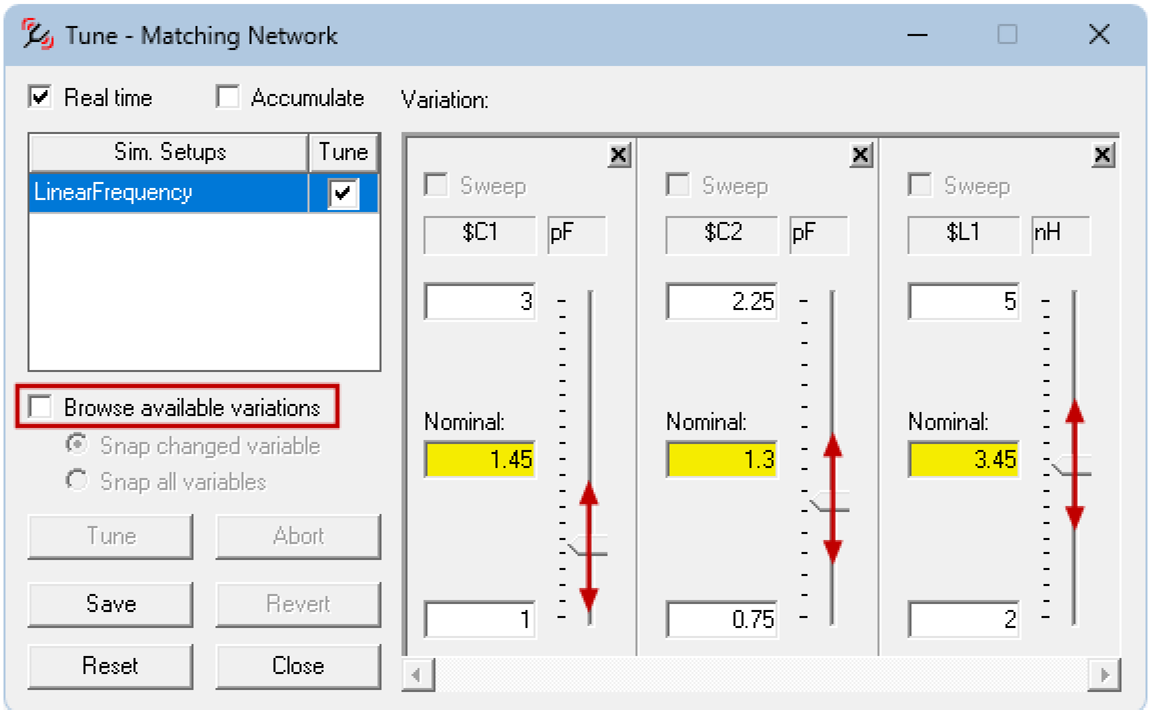
3. Click **OK** to accept the settings and close the *Properties* dialog box.

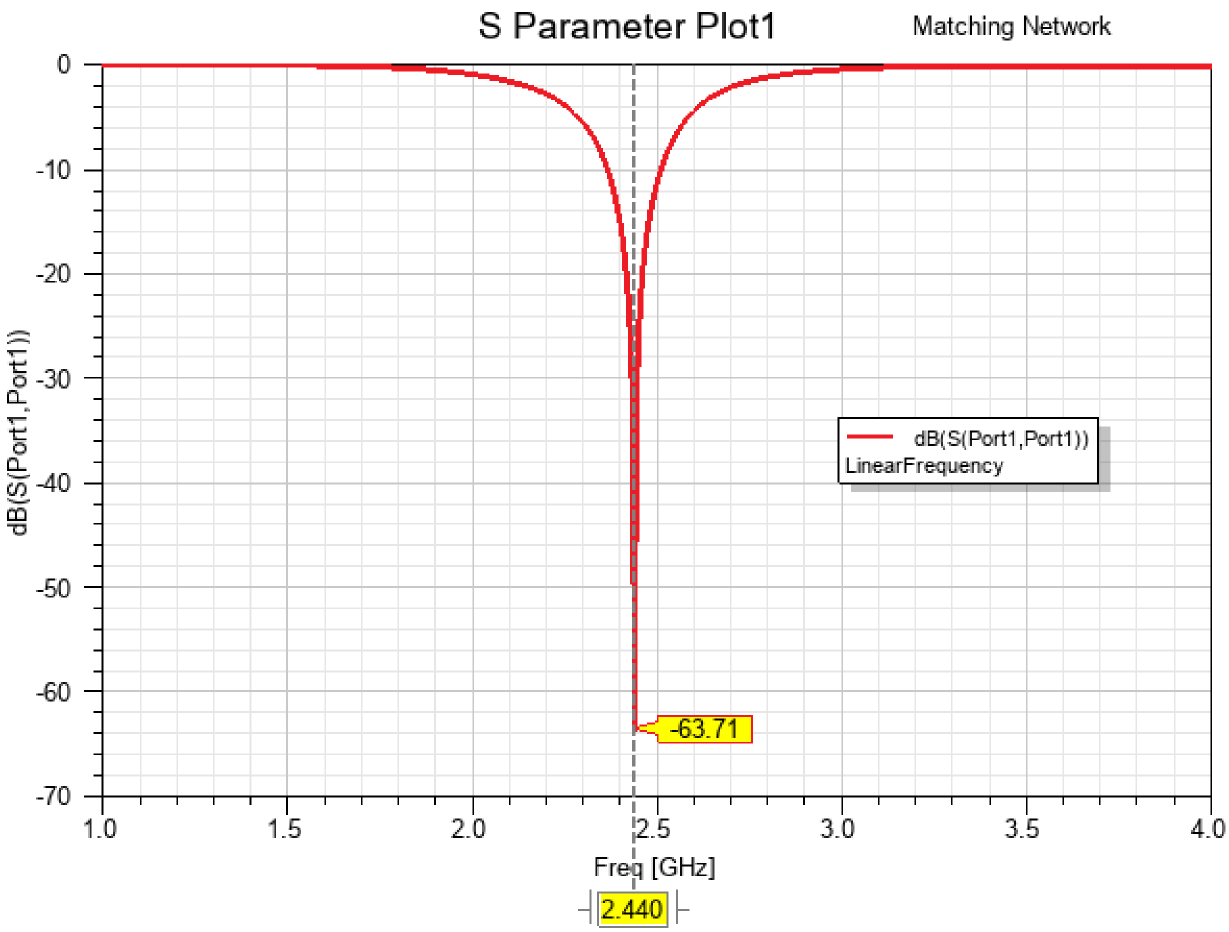
Tune Component Values

In this procedure, you will determine the *$C1, $C2,* and *$L1* component values that provide the minimal S(Port1,Port1) parameter (signal reflection or return loss) at a frequency of 2.44 GHz. Your results may differ somewhat from those shown on this page. Meshing and solution characteristics can vary slightly between different versions of the software and on different platforms. For example, the same component values may result in the minimum return loss at a frequency other than 2.44 GHz, or your decibel value may be somewhat different. Try to achieve a return loss no greater than -40 dB at 2.44 GHz.

1. Bring the **S Parameter Plot 1** window back to the foreground using one of the following methods:
   - If the plot window is partially visible, just click anywhere in the window or on its title bar.
   - Using the **Window** menu, select **Chip_Antenna_Board - Matching Network - S Parameter Plot 1**.
   - Under *Matching Network* > *Results* in the Project Manager, double-click **S Parameter Plot 1**. (This method will reopen the plot window if you previously closed it.)
2. Under *Matching Network* in the Project Manager, right-click **Optimetrics** and choose **Tuning** from the shortcut menu. Alternatively, using the menu bar, click **Circuit** > **Tune**.
3. To provide an unobstructed view of the S Parameter plot, particularly in the area around 2.44 GHz, reposition the *Tune* dialog box that appears. (Do this by clicking and dragging its title bar.)
4. Clear the **Browse available variations** option in the *Tune* dialog box.
5. For each of the three variables in the *Variations* section, slowly drag the **sliding pointers** upward and downward to adjust the values within the *Min* to *Max* range. Allow sufficient time for the plot to update at each component value step. The goal is to minimize the value at the point of intersection of the plot trace and the X marker line. This set of adjustments is tricky because the three components have a complex interaction. You can't alter one variable without affecting the optimal setting for the other two variables.

The *Tune* dialog box and updated *S Parameter Plot 1* should resemble the following two images when you reach the optimal settings:





6. **Close** the *Tuning* dialog box.

   The *Apply Tuned Variation* dialog box appears.

7. Click **OK** to apply the selected variation, which represents the current slider positions.

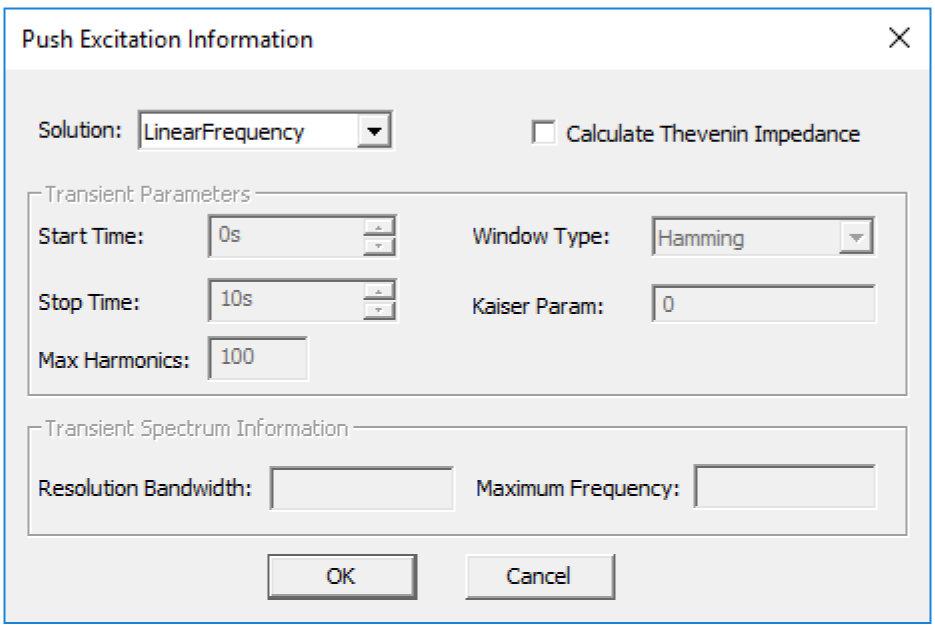
8. **Save** the project.

6 - Push Excitations

Push the excitation information from the Circuit design to the HFSS design as follows:

1. Bring the **Schematic** window back to the foreground using one of the following methods:
    - If the *Schematic* window is partially visible, just click anywhere in the window or on its title bar.
    - Using the **Window** menu, select **Chip_Antenna_Board - Matching Network - Schematic**.
    - Using the menu bar, click **Circuit** > **Schematic Editor**.
    - In the Project Manager, double-click **Matching Network**.
2. In the *Schematic* window, right-click the HFSS design component (**evaluation board**) and choose **Push Excitations** from the shortcut menu.

    The *Push Excitation Information* dialog box appears:

3. Click **OK**.

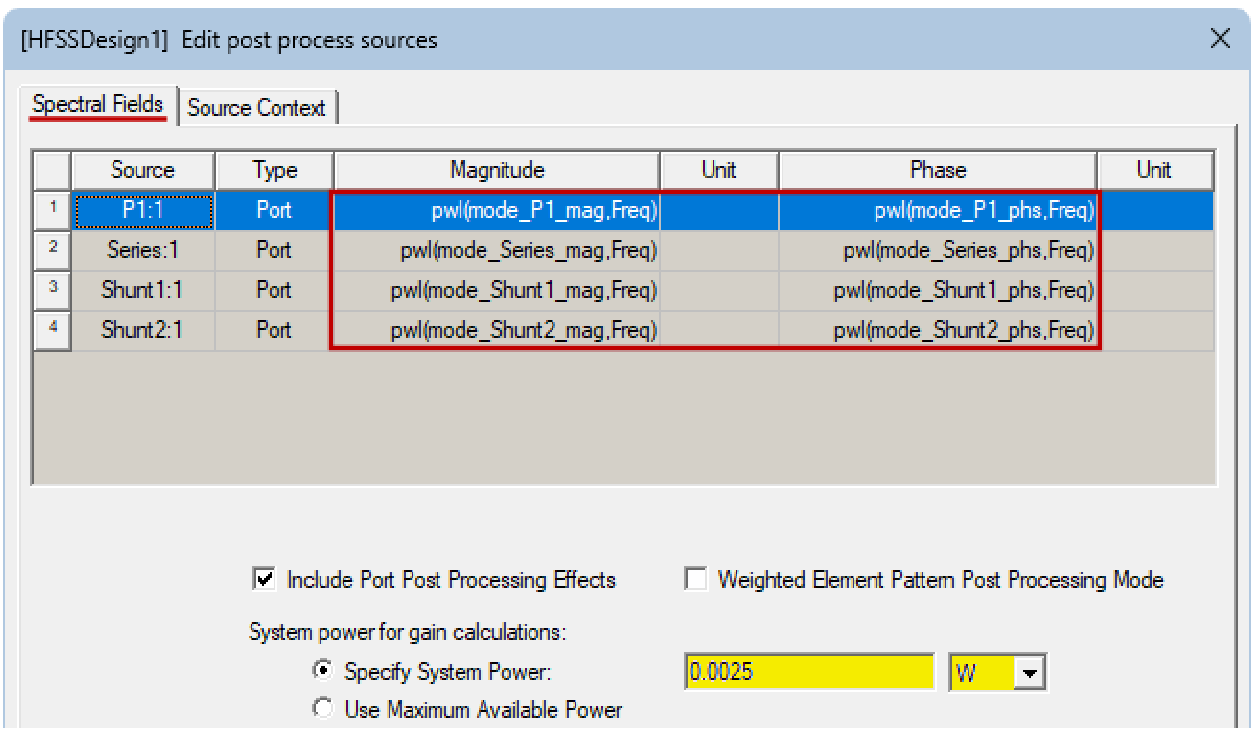
7 - Verify Excitation Data in HFSS

Next, verify that the excitation information has been transferred from the Circuit design to the HFSS design. This information includes the following items:

- Frequency-dependent port excitation magnitude data (four datasets)
- Frequency-dependent port excitation phase data (four datasets)
- System power

1. In the Project Manager, collapse the **Matching Network** branch and expand the **HFSS_ Circuit-Linked (Modal Network)** branch.
2. Double-click **HFSS_Circuit-Linked (Modal Network)** to make it the active design and to bring the Modeler window to the foreground.
3. Under *HFSS_Circuit-Linked (Modal Network)* in the Project Manager, right-click **Excitations** and choose **Edit Sources** from the shortcut menu.

   The *Edit post process sources* dialog box appears.
4. In the *Spectral Field*s tab of the dialog box, ensure that the **Specify System Power** option is selected under *System power for gain calculations*:



The system power value has been updated to reflect the excitation defined in the circuit schematic, as explained below:

- The specified peak AC magnitude is 1 V with a 50 Ω generator impedance.
- The chip antenna board's port impedance is also 50 Ω. Therefore, the voltage divides in half across the two equal series impedances, resulting in a 0.5 V peak magnitude at the HFSS port.
- The system power is $V^2 / 2Z = 0.5^2 / 100 = 0.0025$ W
- The peak current (J) is V / Z = 0.5 / 50 = 0.01 A
- The system power can also be expressed as VJ / 2 = 0.5 * 0.01 / 2 = 0.0025 W

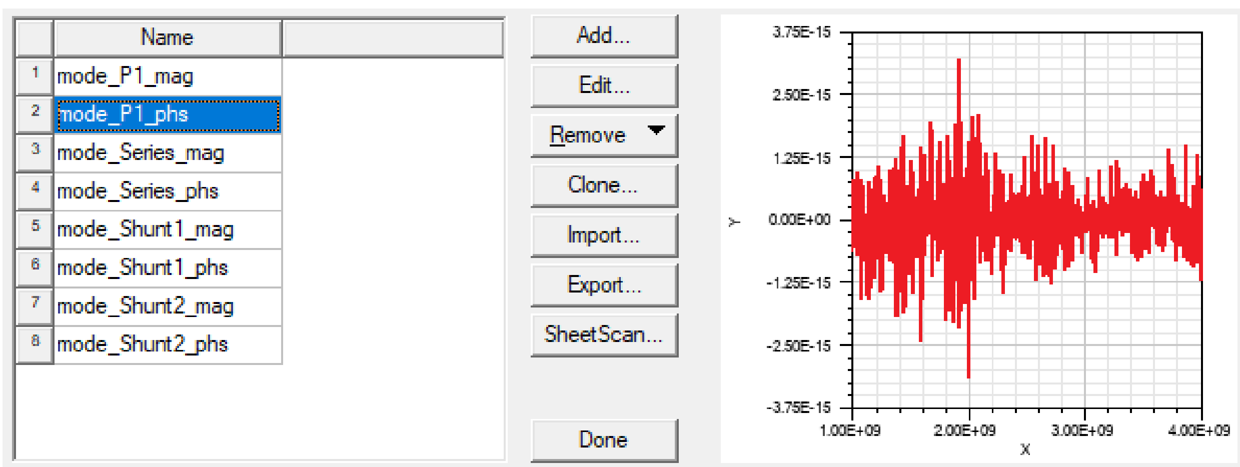
Also, notice that the *Magnitude* and *Phase* columns contain references to frequency-dependent magnitude and phase datasets. Prior to pushing the excitation information from the Circuit design, this dialog box would have shown 1 W magnitude at 0° phase for the first port and all zeros for the magnitude and phase values of the other three ports.

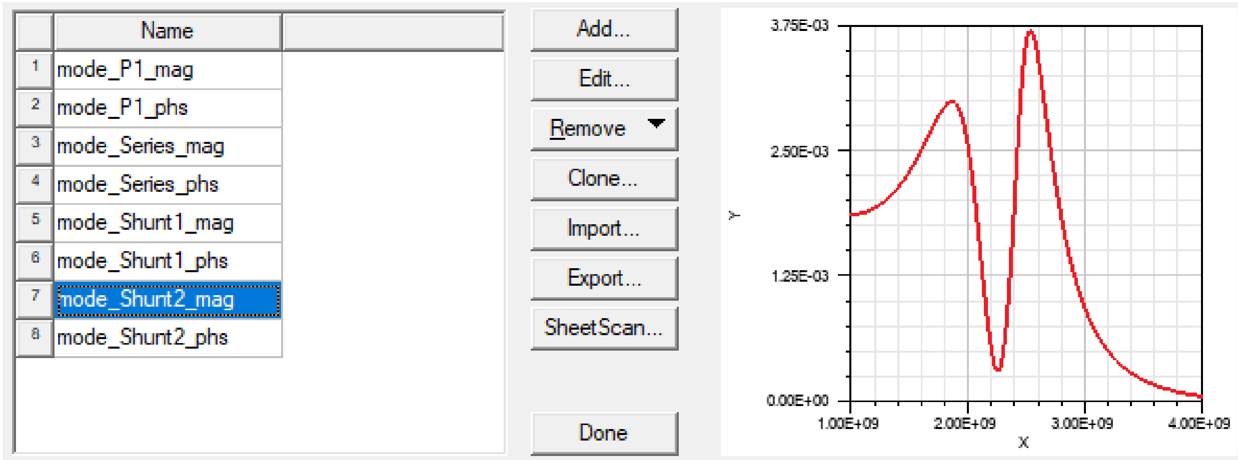
5. Click **OK**.
6. Using the menu bar, click **HFSS** > **Design Datasets**.

   The *Datasets* dialog box appears.

7. Select any of the listed datasets to see a graph of the frequency-dependent magnitude or phase data.

   Two of the eight dataset graphs are shown below:



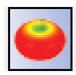
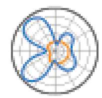

Name
1 mode_P1_mag
2 mode_P1_phs
3 mode_Series_mag
4 mode_Series_phs
5 mode_Shunt1_mag
6 mode_Shunt1_phs
7 mode_Shunt2_mag
8 mode_Shunt2_phs
Add...
Edit...
Remove
Clone...
Import...
Export...
SheetScan...
Done
3.75E-03
2.50E-03
1.25E-03
0.00E+00
Y
1.00E+09
2.00E+09
3.00E+09
4.00E+09
X

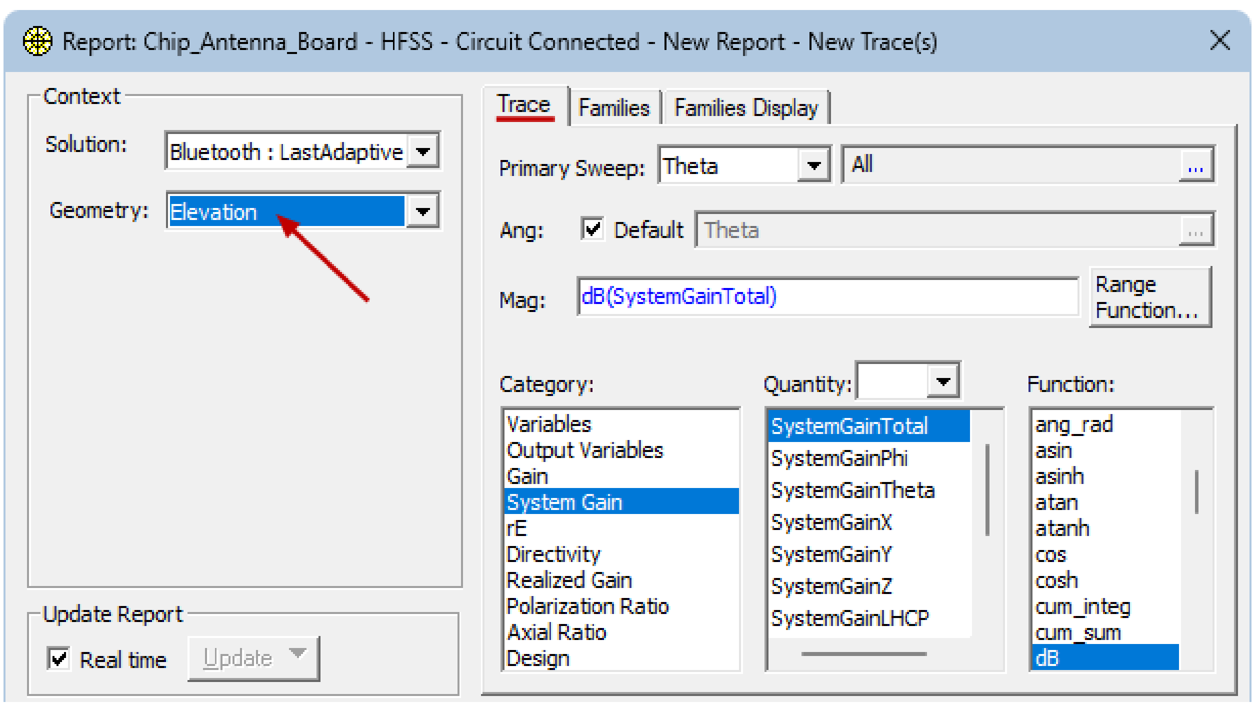
8. Click **Done**.

8 - Create and Overlay Gain Plots

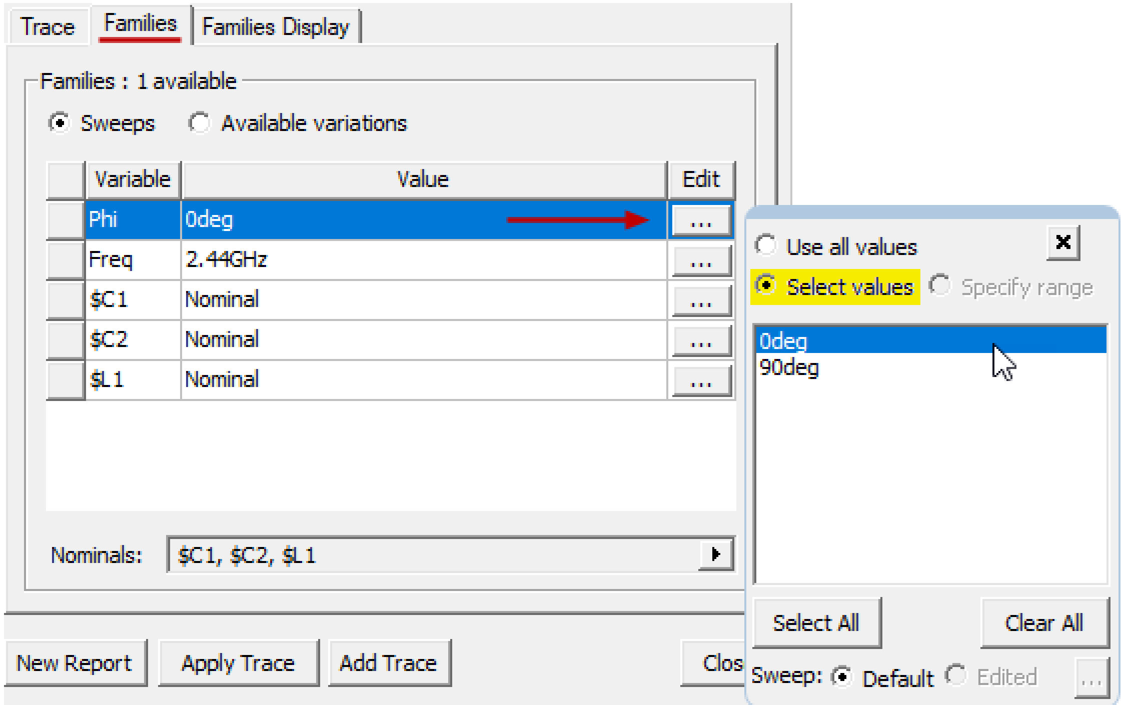
Create two Total System Gain versus Theta plots, one at Phi = 0° and one at Phi = 90°. Gain plots are far fields reports showing the antenna's radiation pattern. You could plot both Phi values as two traces on a single plot. However, after creating the plots, you will overlay the radiation patterns on the model geometry to better visualize the radiation patterns in different directions. To qualify for overlaying, a 2D polar plot cannot contain multiple Phi values. However, you can overlay multiple individual 2D polar plots, each based on a single Phi values.

These System Gain plots will include the effects of the tuning performed in the *Matching Network* (Circuit) design and pushed to the HFSS excitation sources. Create and overlay them as follows:

1. On the **Results** ribbon tab, click  **Far Fields Report** >  **Mag/Ang Polar**.
2. In the *Report* dialog box that appears, specify the following settings:
    a. Select **Elevation** from the **Geometry** drop-down menu.
    b. Ensure that all settings under the **Trace** tab are as shown below:

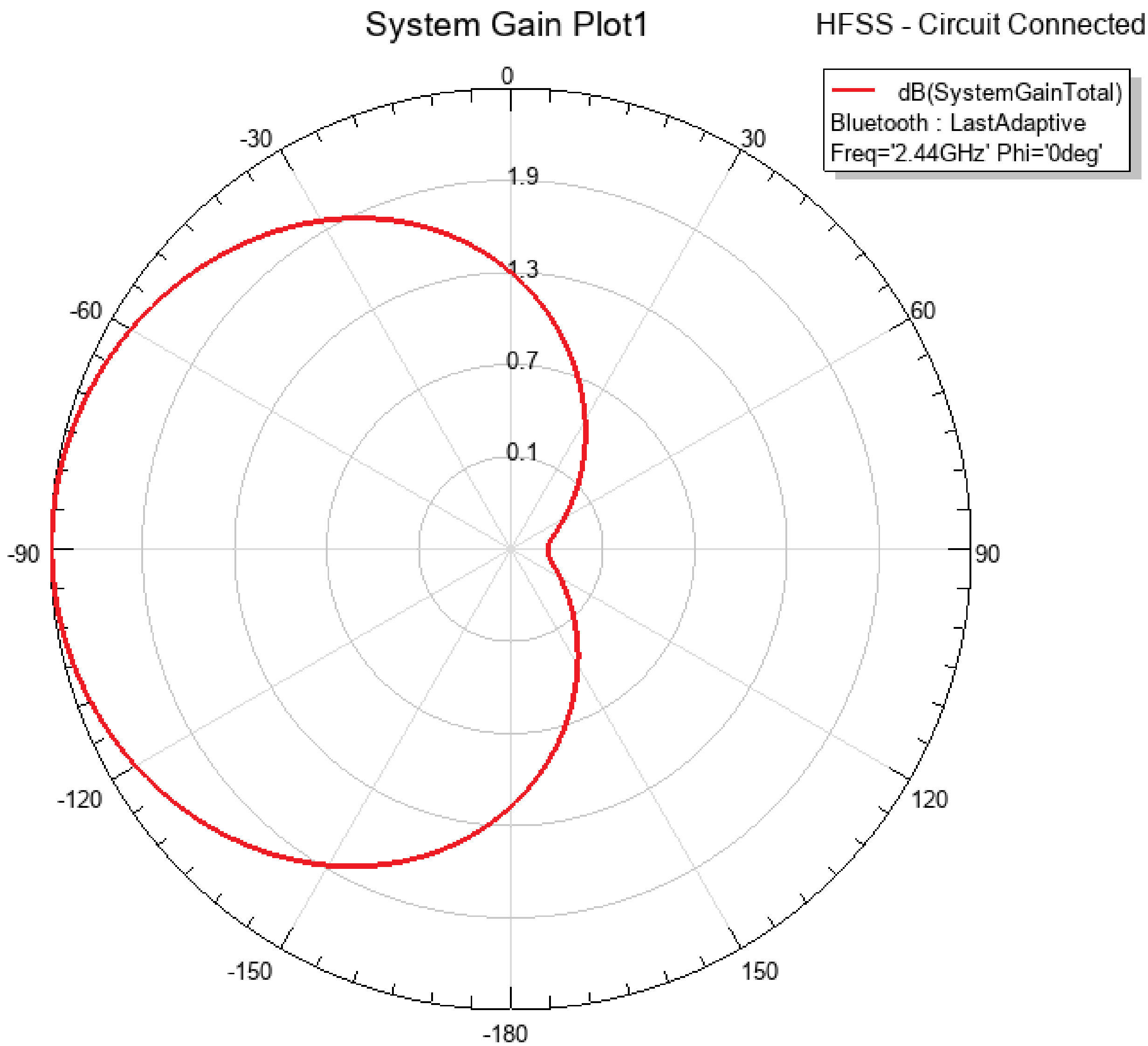


3. Under the **Families** tab, select the **Phi** value of **0deg**, as shown below:



4. Click **New Report** but keep the dialog box open.

   System Gain Plot 1 appears in a new window:

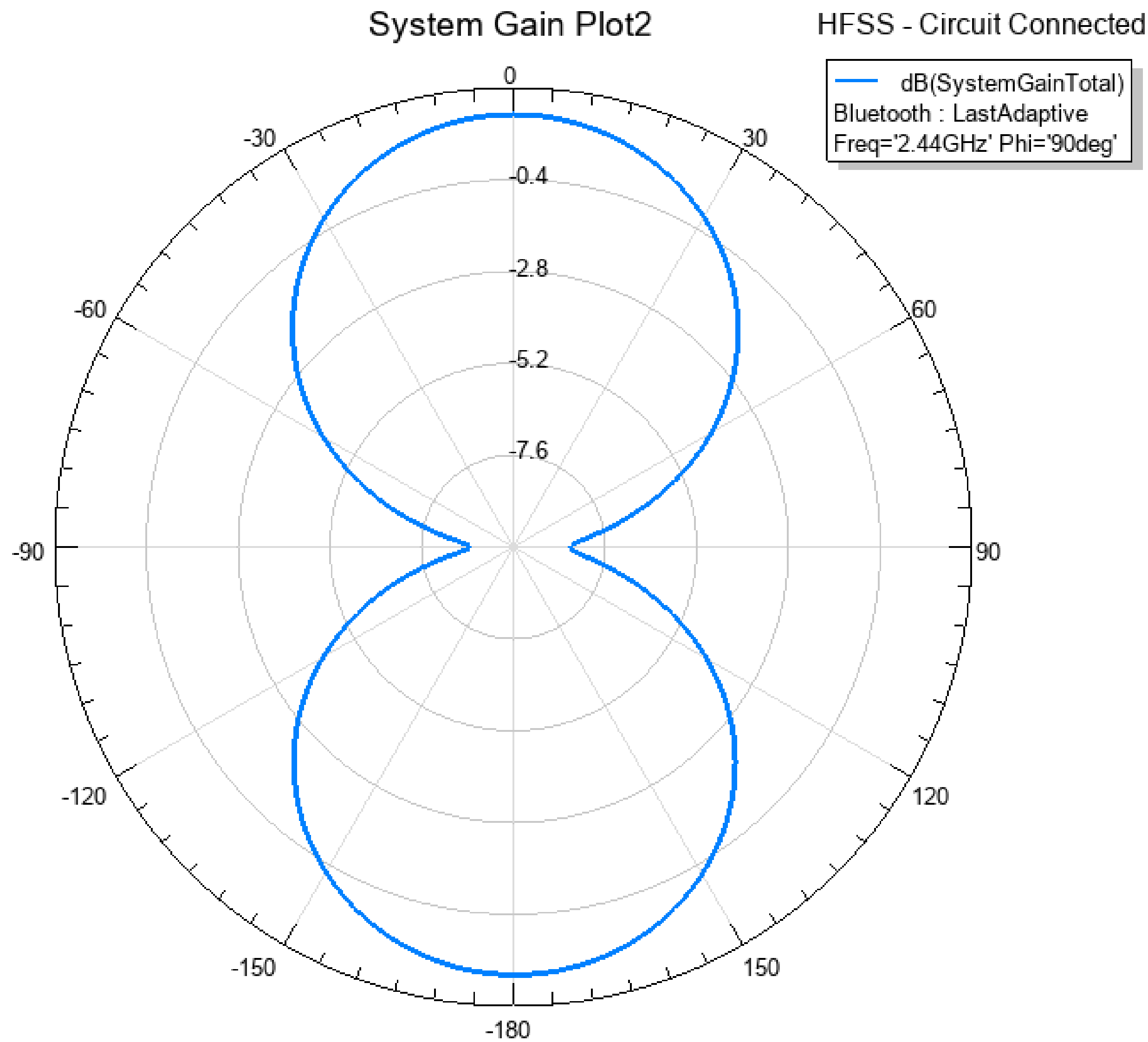


5. In the **Families** tab of the *Report* dialog box, select the **Phi** value of **90deg**.
6. Click **New Report** and click **Close**.

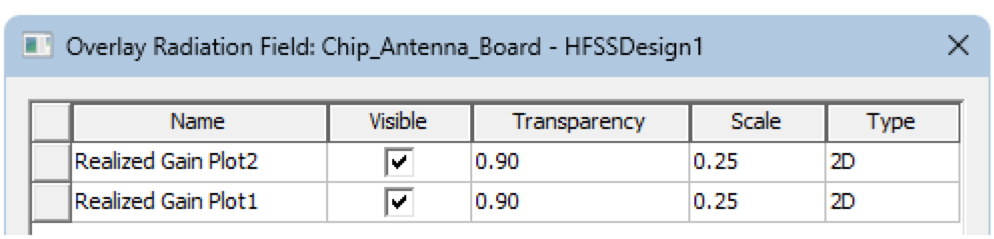
   System Gain Plot 2 appears in a new window.

   To differentiate the appearance of the two gain plot curves, you will next change the color of the trace in *System Gain Plot 2*.
7. Click the trace in *System Gain Plot 2* to select it.
8. In the docked *Properties* window, change the **Color** value from *red* to **medium blue** (column 6, row 1 of the *Basic colors*; Red: 0, Green: 128, Blue: 255) and click **OK**.

   The modified plot should look like the following image:



9. Use the **Window** menu to bring the *Modeler* window to the foreground (**Chip_Antenna_ Board - HFSSDesign*x* - Modeler**).
10. Right-click in the *Modeler* window and choose **Plot Fields** > **Radiation Field**.
11. In the Overlay radiation field dialog box that appears, specify the following settings:

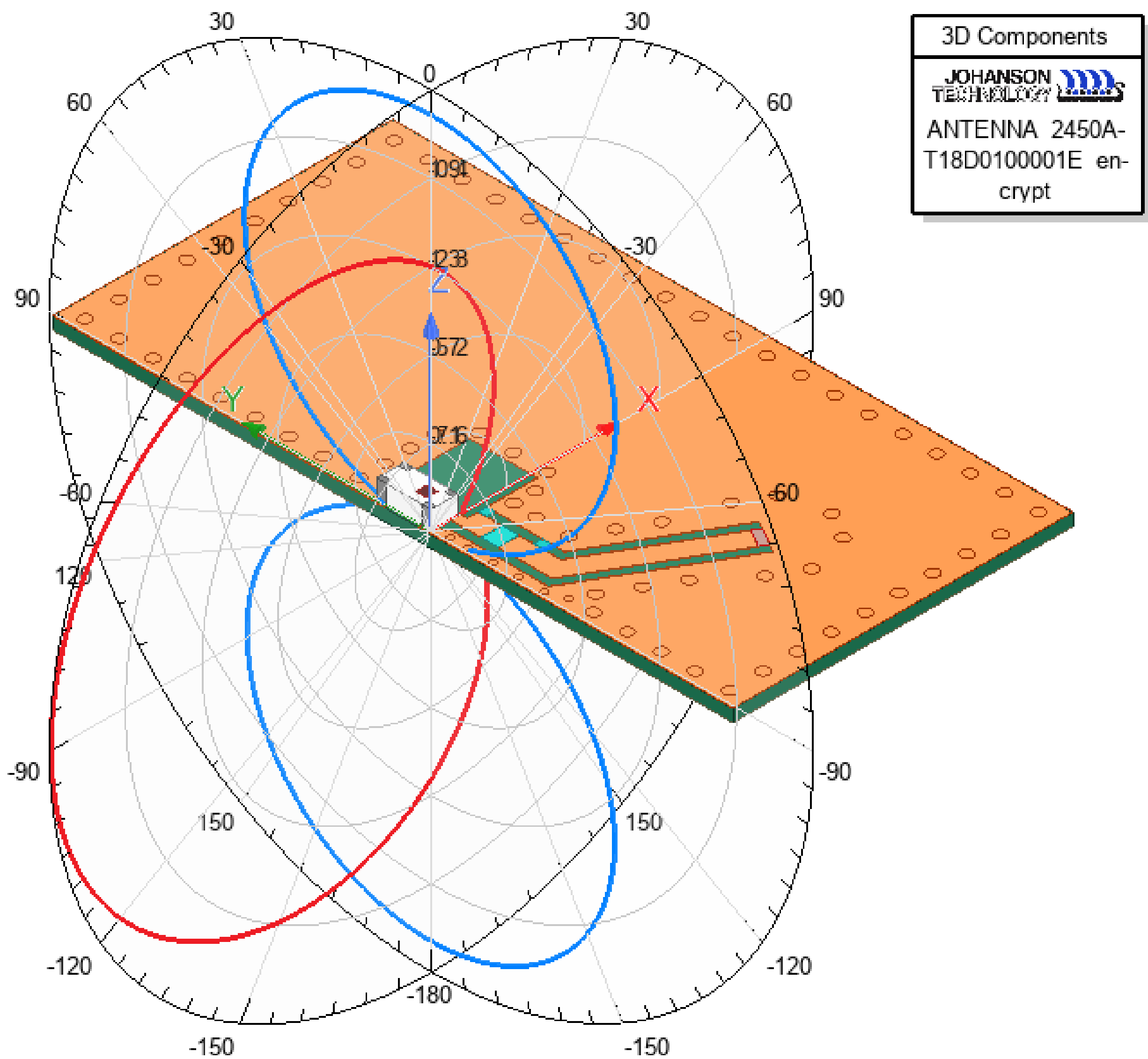

Overlay Radiation Field: Chip_Antenna_Board - HFSSDesign1

Name	Visible	Transparency	Scale	Type
Realized Gain Plot2	☑	0.90	0.25	2D
Realized Gain Plot1	☑	0.90	0.25	2D

12. Click **Apply** and then **Close**.
13. With the cursor near the upper-left corner of the Modeler window's display area, press and hold **Alt** while **double-clicking** the left mouse button. This action produces an alternative isometric view of the model with the back side facing towards you.

> **Note:**
>
> This viewpoint prevents a significant portion of the red and blue traces from being hidden by the evaluation board.

The overlaid plots should resemble the following image:



14. **Save** the project.

You have completed the *Matching Network* getting started guide. Simulating manual tuning of the network is useful because it gives you a good idea about the complex interactions of the actual tuning components and how they behave when performing an alignment on the workbench. Again, if you would like to see how to automatically optimize the matching network tuning, either using a linked Circuit design or doing so entirely within HFSS alone, please refer to the Johanson_2450AT18D0100-EB1SMA example model.