Ansys 2024 R2

Take A Leap Of Certainty™

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

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 2024 R2
July 2024

Conventions Used in this Guide

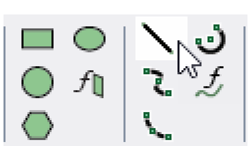
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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

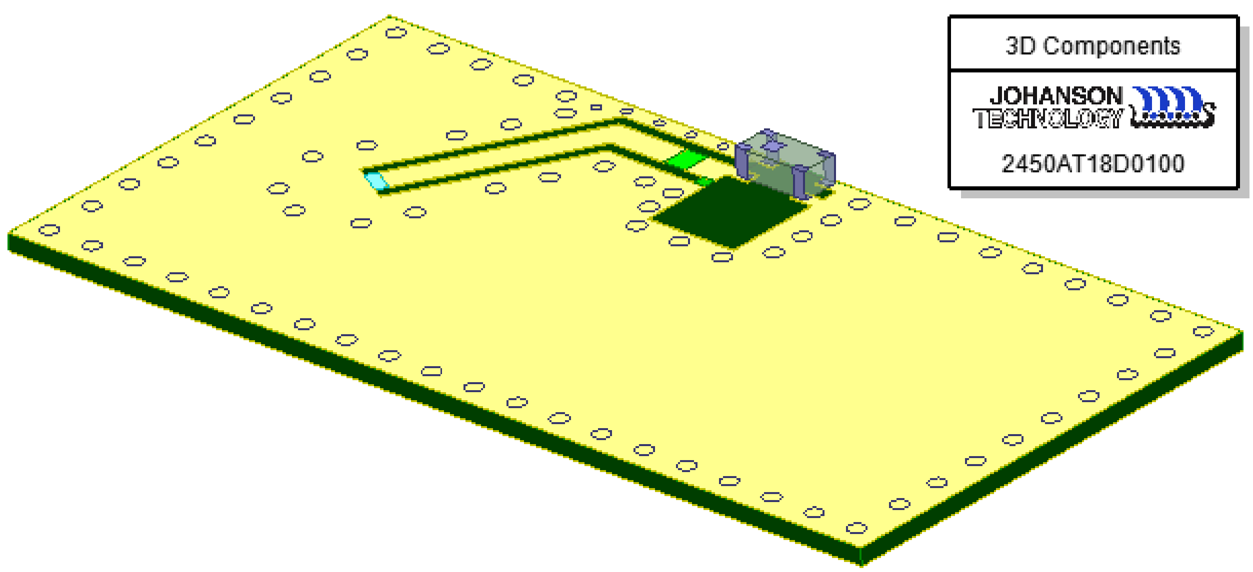
In this *Getting Started* guide, you will learn about dynamic linking capabilities between *HFSS* and *Circuit* designs in the *Ansys Electronics Desktop* application. Specifically, the guide provides an example of a matching network for a Bluetooth chip antenna.

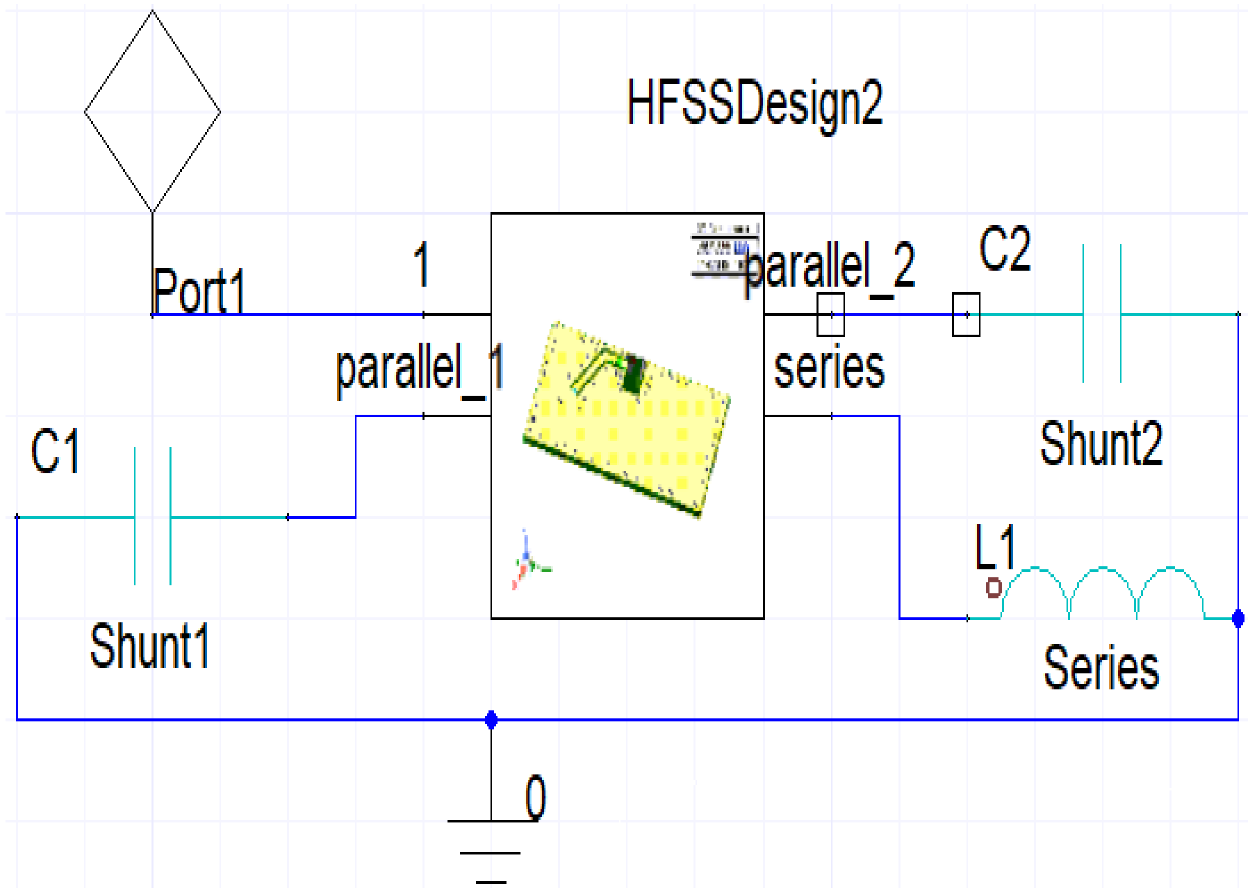
By following the procedures in this guide, you will learn how to perform the following tasks:

- 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 an 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 2450AT18D0100 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* subfolder). However, you will use an incomplete version of this model as your starting point for this exercise.

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




HFSSDesign2
Port1
1
parallel_1
parallel_2
C2
series
C1
Shunt2
L1
Shunt1
Series
0

2 - Set Up the Project

In this chapter, you will perform the following tasks:

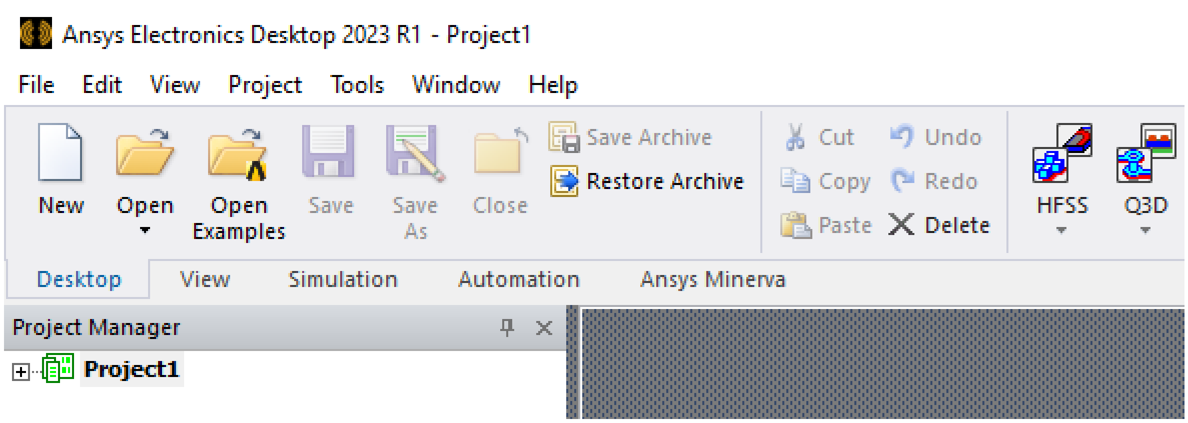
- Launch the Ansys Electronics Desktop application
- Open the evaluation board project file
- Enable legacy view orientations
- Insert the chip antenna from the *Components Libraries*
- Rename the predefined lumped port excitation

Launch Ansys Electronics Desktop

For convenience, a shortcut to the Ansys Electronics Desktop (EDT) application is placed on your desktop during program installation. Optionally, you may want to pin the shortcut to your Windows Start Menu too. Before proceeding to the next topic, launch EDT and add a blank project, as follows:

1. Double-click the EDT **Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu).

   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.

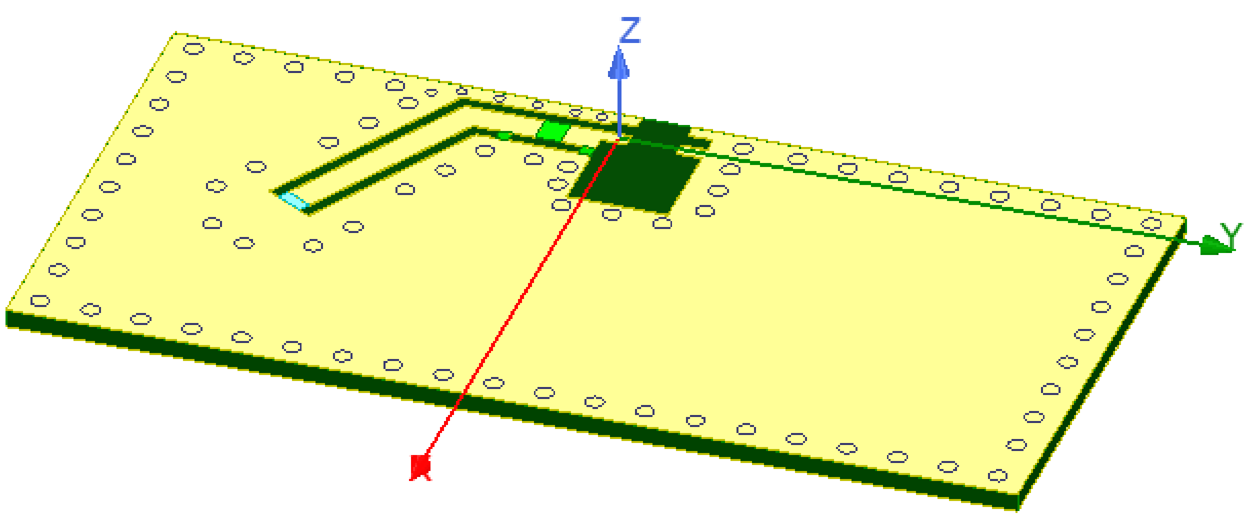
2. Right-click **Project*x*** at the top of the Project Manager and choose **Close** from the shortcut menu.

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 this reason, the project file is located in the *Help* folder instead of the *Examples* folder.

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.

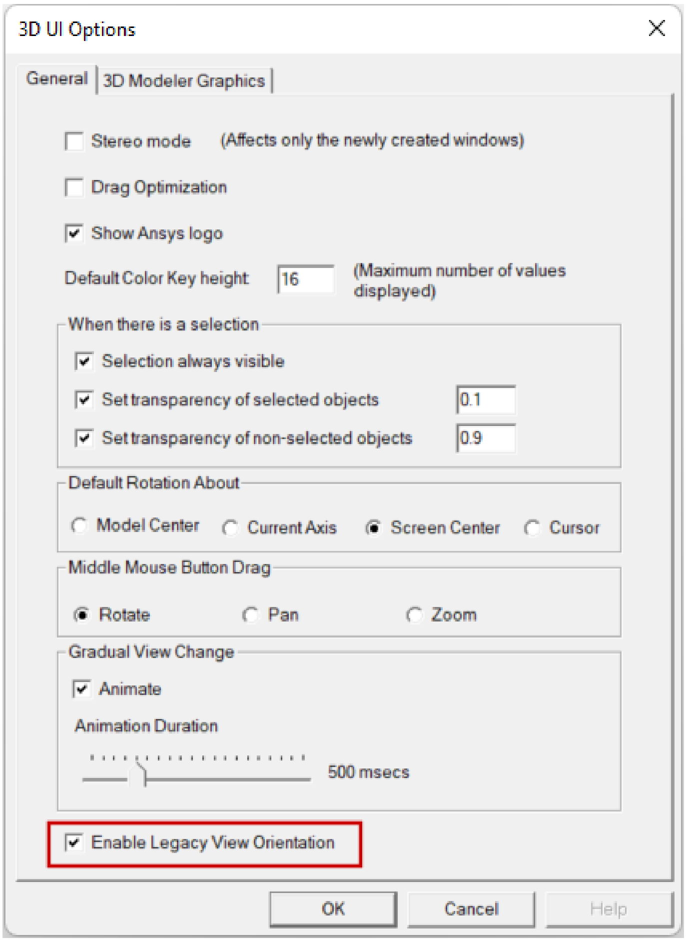
Enable Legacy View Orientations

This getting started guide was created based on standard view orientations that were in effect for version 2023 R2 and earlier of the Ansys Electronics Desktop application. For consistency between your experience and the views and instructions contained in this guide, select the *Enable Legacy View Orientation* option in the 3D UI Options dialog box, as follows:

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

   The *3D UI Options* dialog box appears.

2. Select **Enable Legacy View Orientation**:



3. Click **OK**.

   Changing the view orientation option does not change the model viewpoint that was in effect at the time.

4. On the **Draw** ribbon tab, click **Orient** to change to the *Trimetric* view, which is the default legacy view orientation.

   You do not have to select *Trimetric* from the *Orient* drop-down menu. The default view appears when you click *Orient*.

Although this option can only be accessed once a design is added to a project, it is a global option. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

At the end of this guide, you will be prompted to clear the *Enable Legacy View Orientation* option, if you prefer to use the view orientation scheme implemented for 2024 R1 and newer versions going forward.

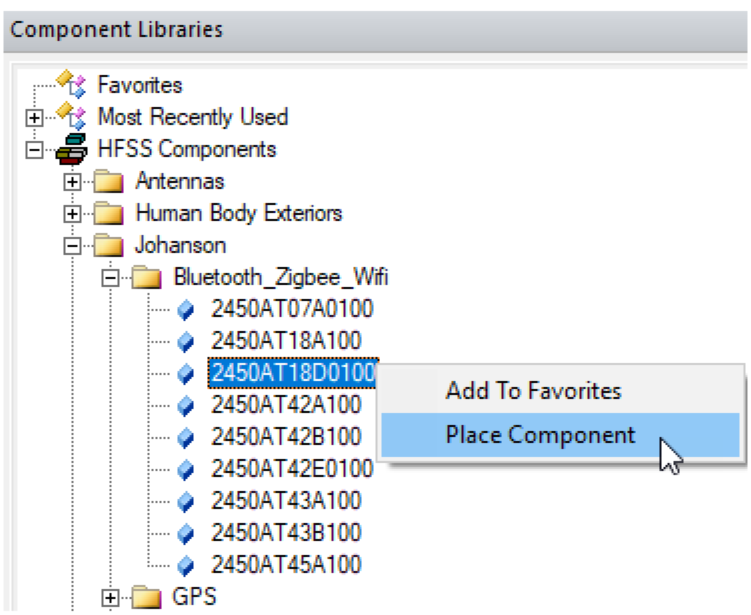
For a comparison of the legacy and current view orientations, search for *"View Options: 3D UI Options"* in the HFSS help. Additionally, views associated with **Alt + double-click** zones have been redefined. The current orientations are shown in the help topic, *"Changing the Model View with Alt+Double-Click Areas."*

Insert Chip Antenna

The chip antenna evaluation board has a coordinate system predefined (*AntCS*), which has its origin specified at the antenna insertion point. For this particular model, that point corresponds to the global origin, and the *Global* and *AntCS* coordinate systems are identical. However, that is often not the case. You will use the predefined AntCS coordinate system to add the antenna to the board, demonstrating the required method for cases in which the antenna insertion point does not correspond to the global origin.

> **Note:**
>
> The Johanson evaluation board model is protected. You cannot move, copy, delete, or otherwise modify any of the objects that comprise it. However, you can add objects to the model.

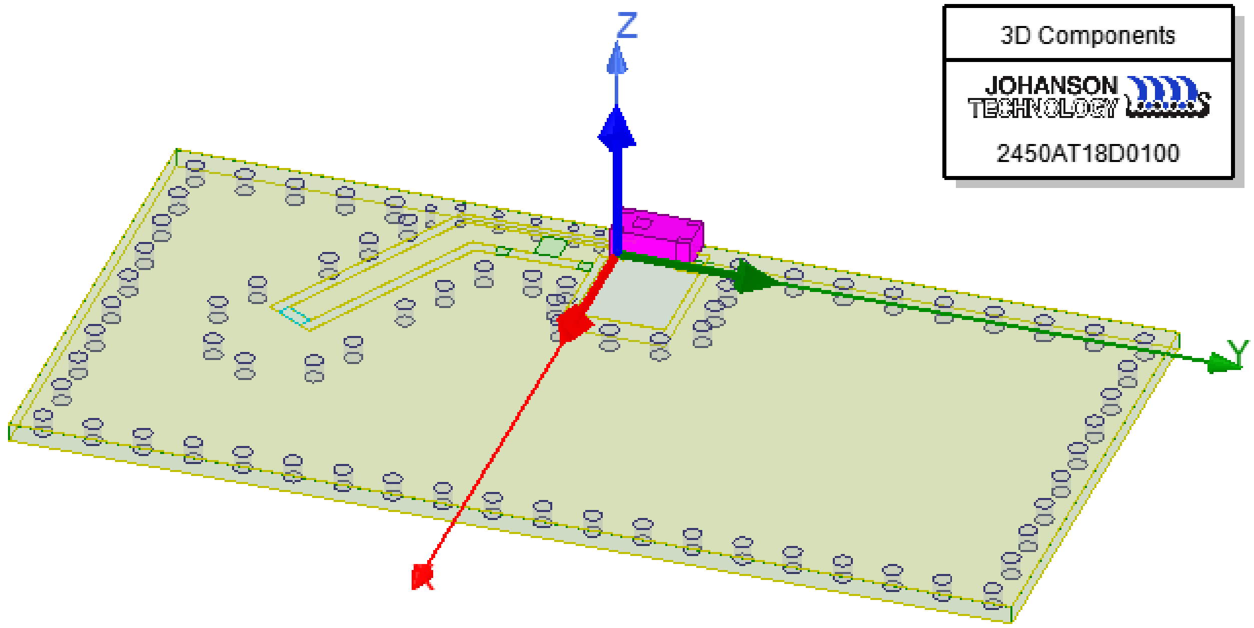
1. Under *Coordinate Systems* in the History Tree, ensure that **AntCS** is the working coordinate system. (A red "W" appears in the lower right corner of the icon when it is active.) If not, select **AntCS** to make it active.
2. If the *Component Libraries* window is not already visible, use the menu bar to toggle its visibility. (Click **View** > **Component Libraries**.)
3. Expand the **HFSS Components** > **Johanson** > **Bluetooth_Zigbee_Wifi** folder in the *Component Libraries* window.
4. Right-click **2450AT18D0100** and choose **Place Component** from the shortcut menu:



The *Insert 3D Component* dialog box appears.

5. Ensure that the **AntCS** is selected from the **Target Coordinate System** drop-down menu.
6. Click **OK** to place the antenna on the board.

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



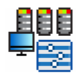
> **Note:**
>
> Alternatively, you could click and drag the antenna from the library to the Modeler window, bypassing the *Insert CD Component* dialog box. You can drop the component anywhere in the Modeler window. It will be automatically positioned at the correct insertion point (the origin of the currently active coordinate system) and correctly oriented.

7. Click in the Modeler window's background area to clear the selection.
8.  **Save** your project. (This command is available from all ribbon tabs and from the *File* menu.)

Set HPC Analysis Options

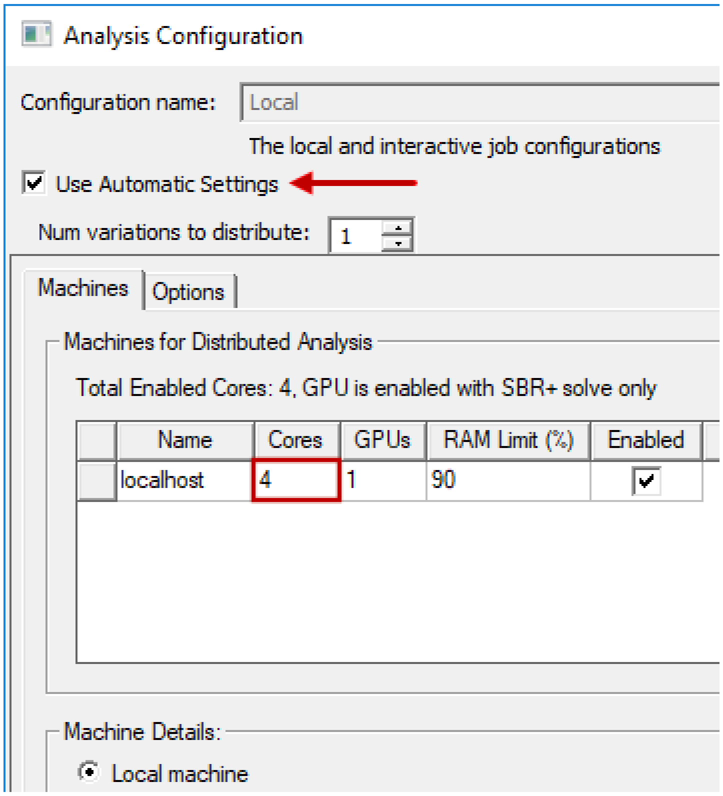
The high performance computing (HPC) capabilities of Ansys Electronics Desktop enable you to take advantage of multiple processing cores, which are common on all high-end computer workstations. Different portions of the analysis can thereby be solved in parallel, shortening the solution time.

> **Note:**
>
> The 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.

3 - Validate and Analyze HFSS Design

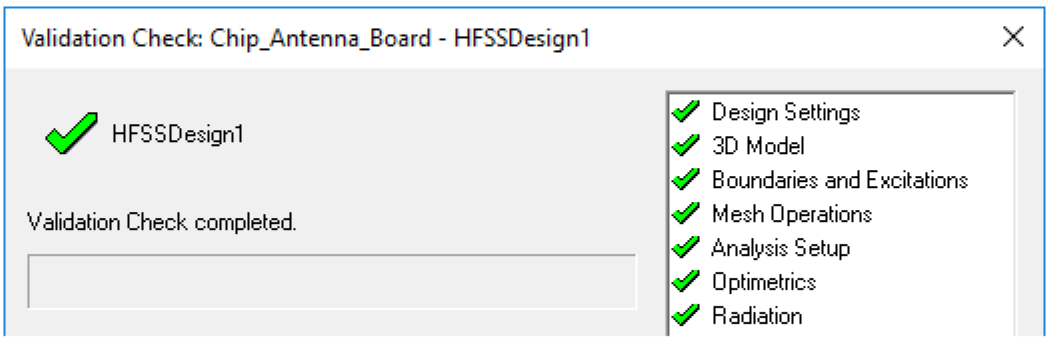
An analysis setup named *Bluetooth* and a 1 to 4 GHz frequency sweep (*1_4GHz*) are already defined in the HFSS design. If desired, you can double-click either of these two entries under *Analysis* in the Project Manager to review the settings. Click **Cancel** in each dialog box to close them without modification before proceeding further.

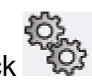
Additionally, an open region, radiation boundary, ground boundary, and four lumped port excitations are already defined. Do not modify any of the boundaries or excitations.

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 less than 15 minutes to solve 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 the HFSS design to 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, right-click **Circuit1** and choose **Rename**.
5. Type **Matching_Network** and press **Enter**.

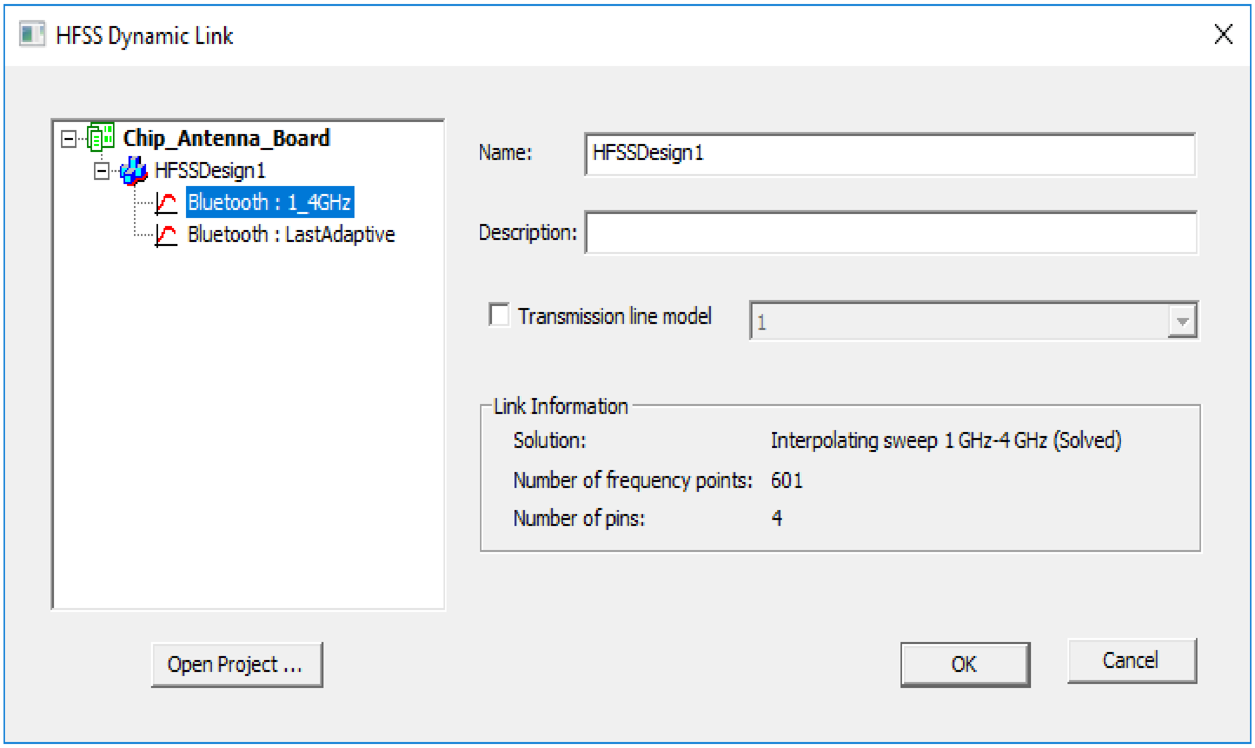
Add HFSS Model to Circuit

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.

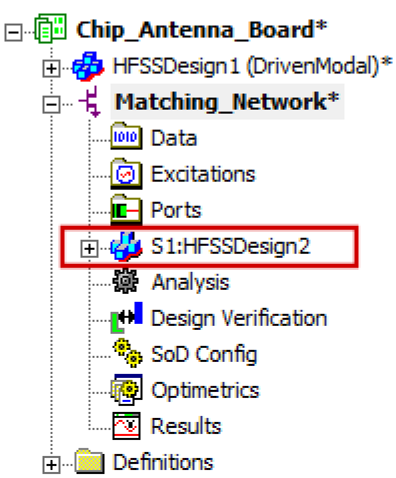
   The *HFSS Dynamic Link* dialog box appears.

2. On the left side of the dialog box, under *Chip_Antenna_Board* > *HFSSDesign1*, select the frequency sweep (**Bluetooth : 1_4GHz**):



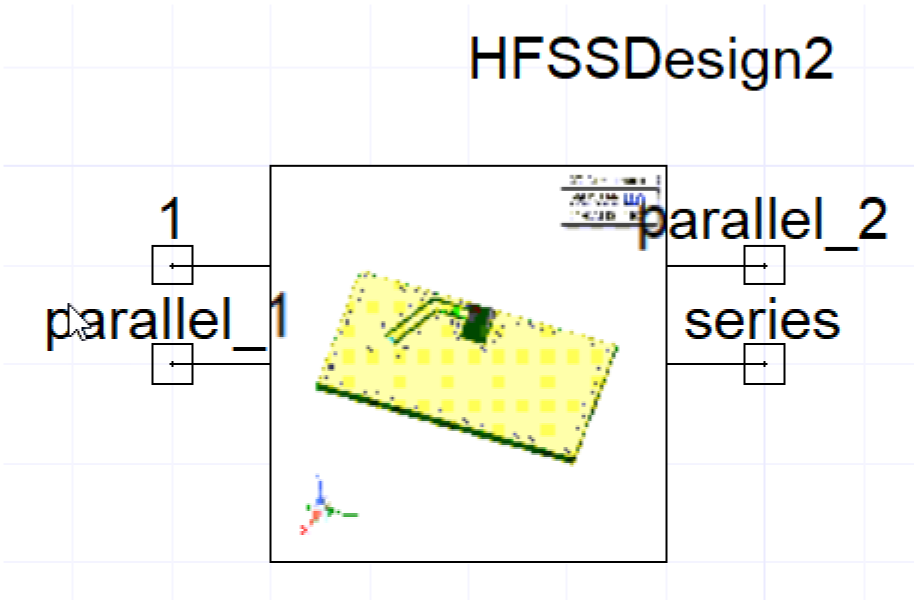
3. Click **OK**.

   A subcircuit model of the HFSS design is inserted into the Circuit design *Matching_Network* in the Program Manager. The number at the end of the HFSS design name has been incremented:



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

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

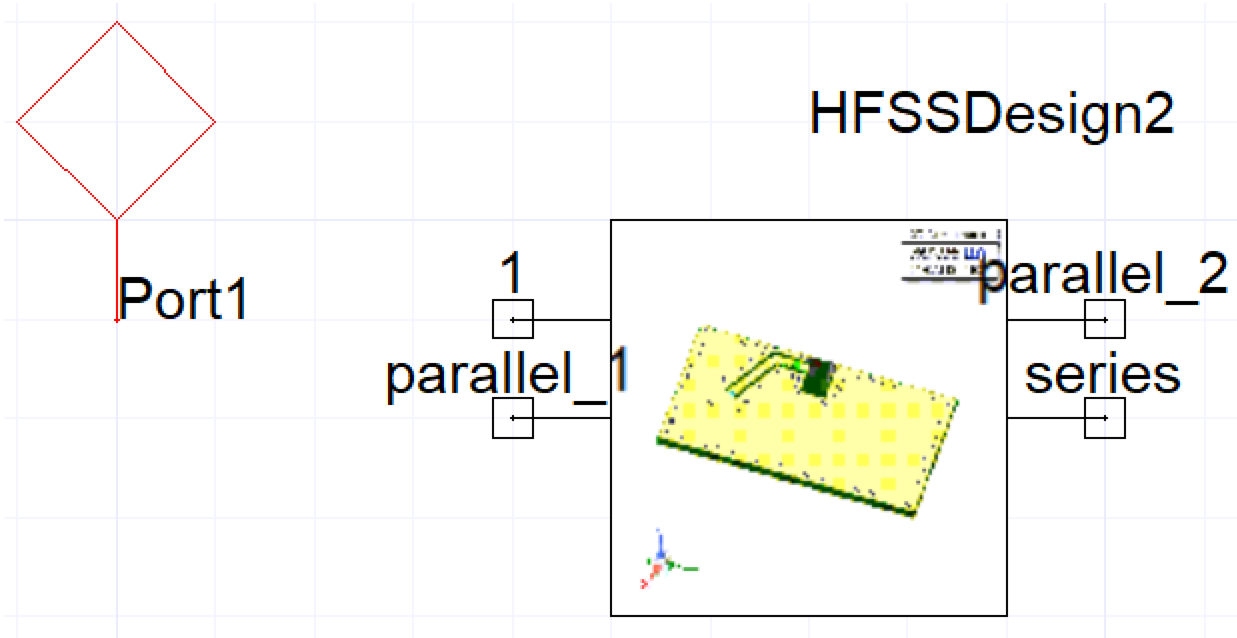


> **Note:**
>
> Zoom in or out and scroll in the Schematic window as needed while completing the circuit. The mouse buttons and scroll wheel work somewhat differently in the *Schematic* window than they do in the 3D Modeler. Roll the mouse wheel to zoom in or out, as in the 3D Modeler. Press **Shift** 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

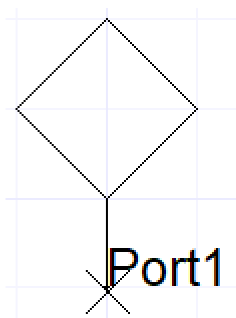
Add an interface port to the schematic. Then, connect it to pin 1 of the HFSS evaluation board model you added to the schematic in the last procedure.

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).

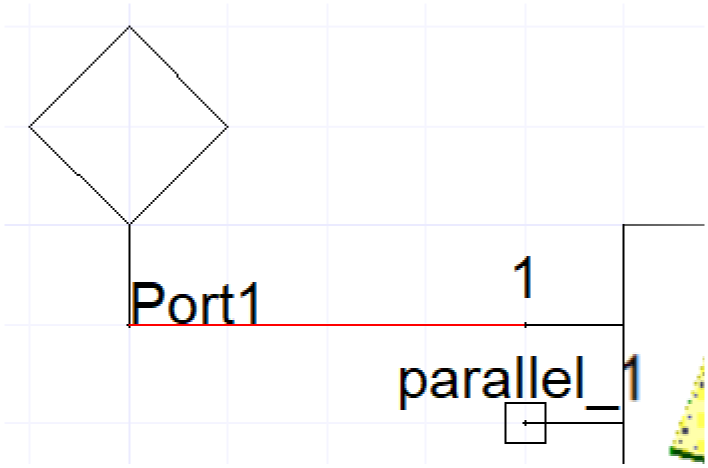
   The cursor changes to an X when near a valid connection point.



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:

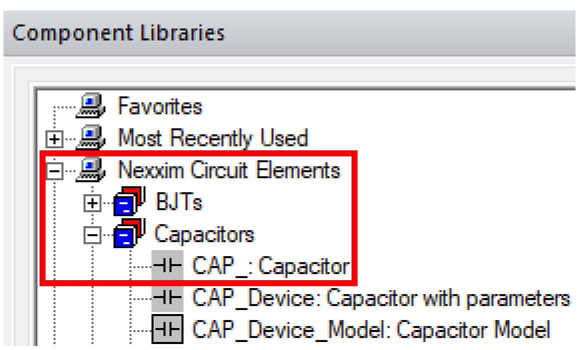


Because the line completes a connection to a pin, the segment is terminated. Otherwise, if you clicked at a non-connection point (say to change directions) you can continue to draw line segments until a valid termination point is reached.

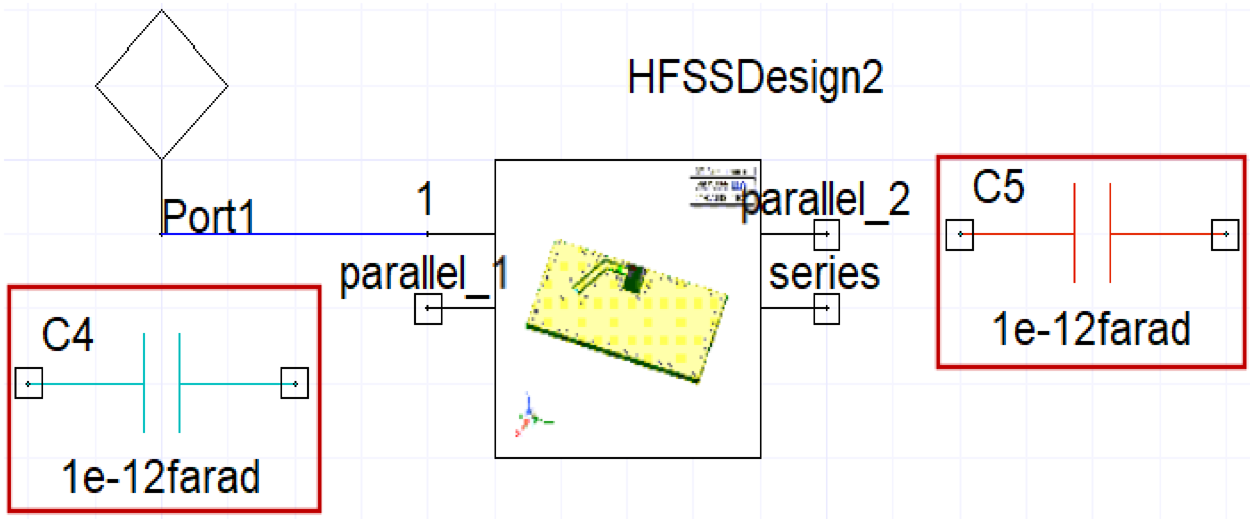
Add Components to Schematic

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.

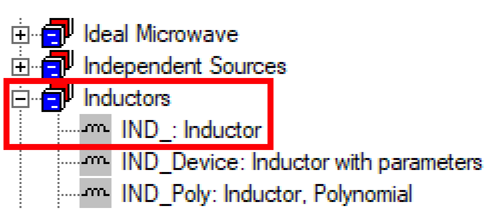


3. Click and drag the component labeled **CAP_: Capacitor** onto the left side of the schematic. Then move the cursor and click again to place a second one on the right side. Locate the two capacitors at the approximate positions shown below:



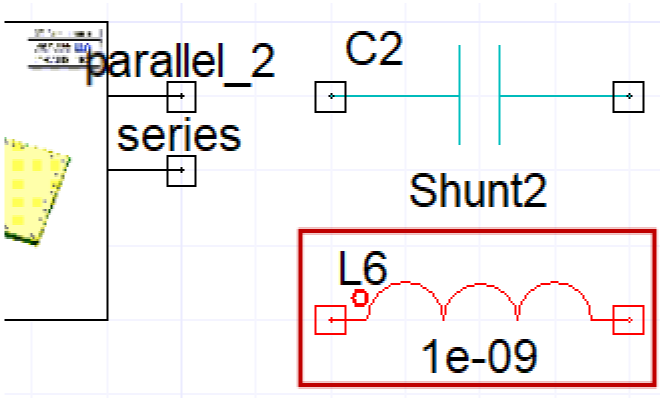
4. Right-click and choose **Finish** (or press **Space** or **Esc**) to terminate the component placement operation.
5. Click the capacitor on the left side of the schematic to select it.

   The settings associated with this component appear in the docked *Properties* window.
6. In the **Param Values** tab of the docked *Properties* window, change the default numerical value of **C** to the variable **Shunt1** 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.
   b. Select **pF** from the **Units** drop-down menu.
   c. Type **1** 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.
9. Repeat steps 5 through 8 for the capacitor on the right side of the schematic. This time, specify the variable **Shunt2** with the same value (**1 pF**) and name it **C2**.
10. Under *Nexxim Circuit Elements*, expand the **Inductors** branch.



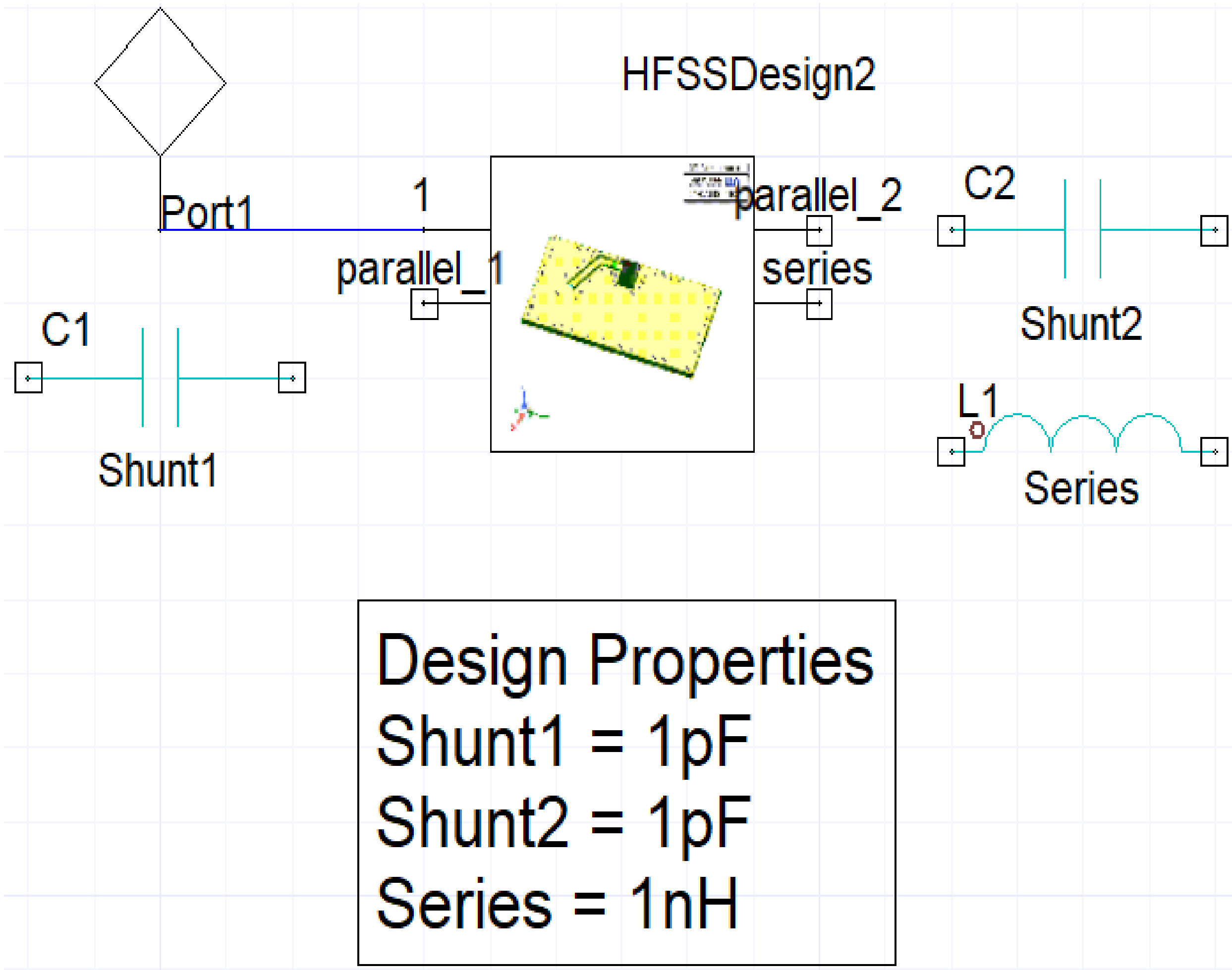
11. Double-click the component labeled **IND_: Inductor**.

12. Right-click at the desired inductor insertion point (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, change the default numerical value of **L** to the variable **Series** 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.
    b. Select **nH** from the **Units** drop-down menu.
    c. Type **1** 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* legend to your preferred location.
18. Click in the background to clear the current selection.

    The schematic should resemble the following image:

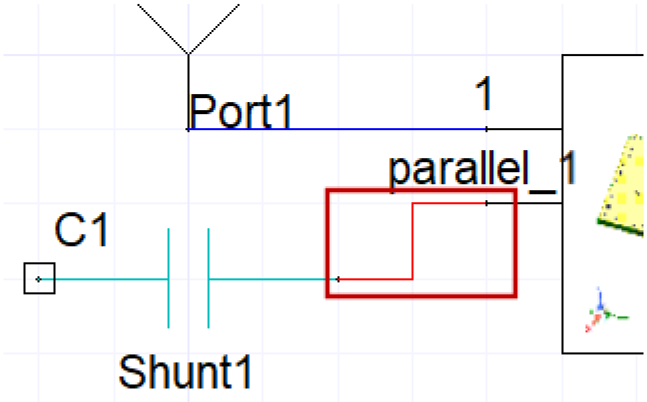


Connect Components in Schematic

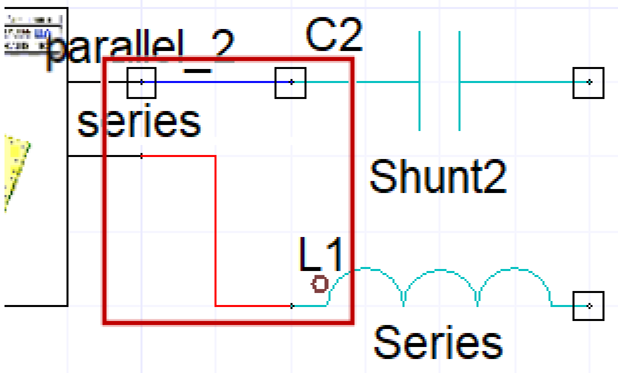
Next, you will connect the discrete components just added to the schematic to the appropriate points on the evaluation board.

1. Click the **right** end of capacitor **C1** to start a connection line, click two intermediate points, and click pin **parallel_1** to terminate the connection.

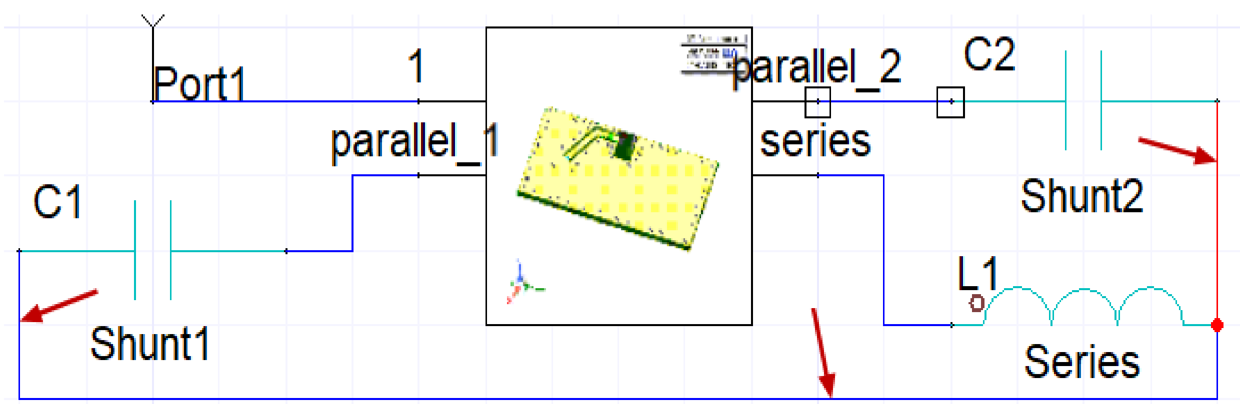
> **Note:**
>
> In a proper schematic, diagonal connection lines are avoided. The intermediate points enable a multi-segment connection in which all line segments are orthogonal. This connection should resemble the following image when completed:



2. In the same manner, connect the **left** end of **C2** to pin **parallel_2** and the **left** end of **L1** to pin **series**:



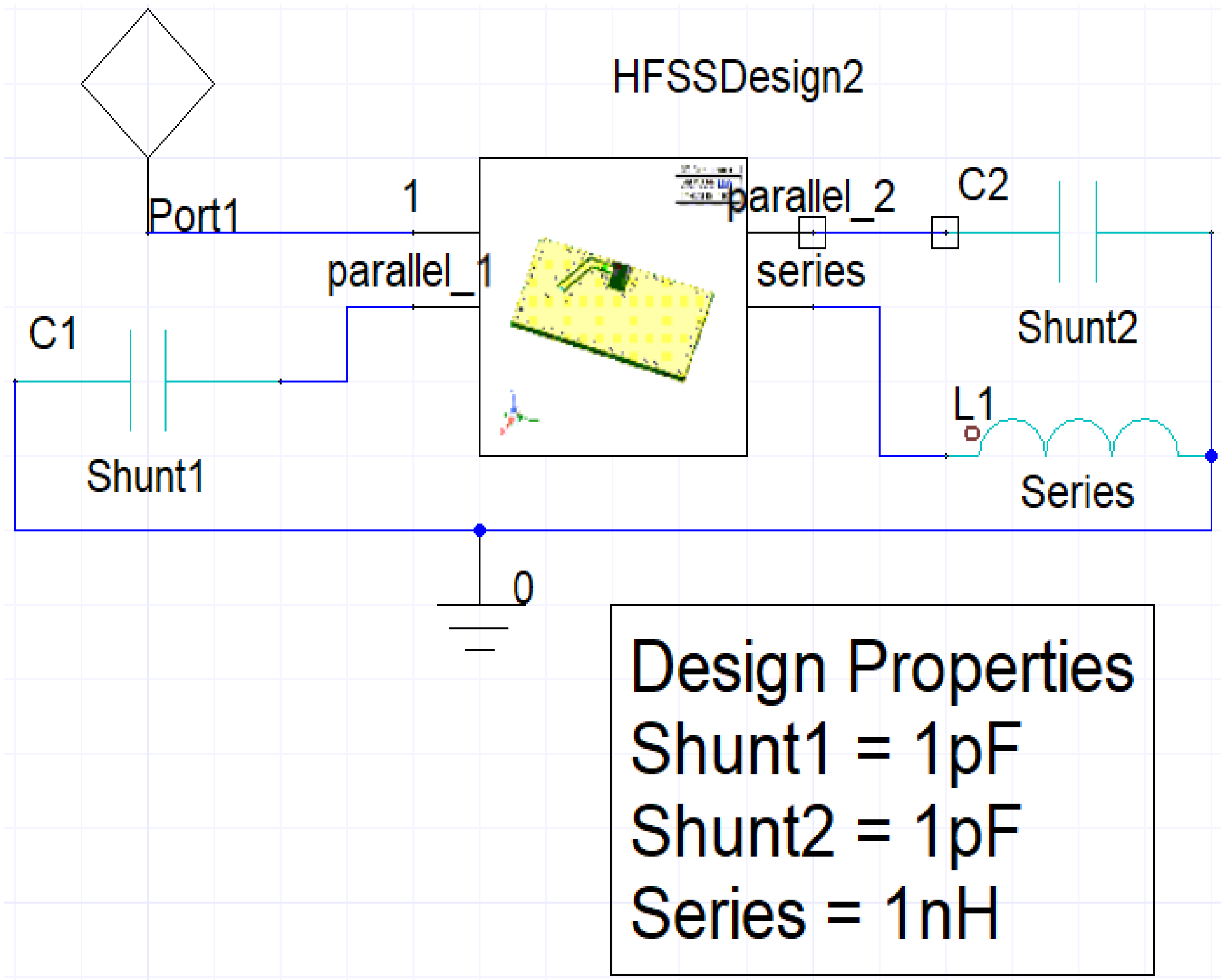
3. Connect the **left** end of **C1** and the **right** ends of **C2** and **L1** to each other, as shown below:



4. On the **Schematic** ribbon tab, click **Ground**.

5. Click to connect the Ground to any grid point along the connection made in step 3.
6. Press **Space** to exit the *Ground* command.
7. Click in the background to clear the selection.

   The finished schematic should resemble the image below:



8.  **Save** the project.

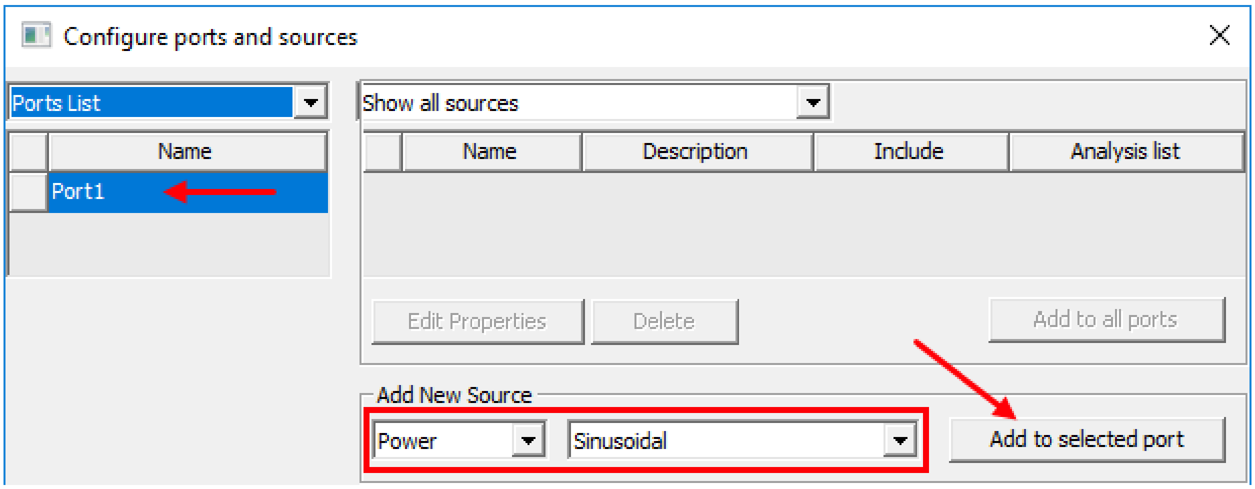
Configure Circuit Excitation

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.

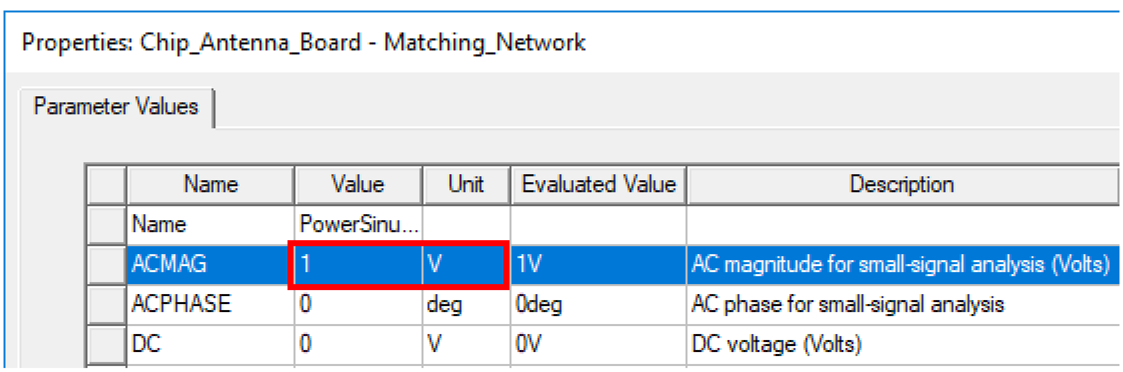
   The *Configure ports and sources* dialog box appears. Port1 is already selected because it's the only item in the Ports List.

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**:

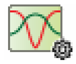


> **Note:**
>
> This voltage is the peak magnitude of the AC signal, not peak-to-peak. The peak-to-peak amplitude = 2 * ACMAG, and the average amplitude = ACMAG / √2.

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

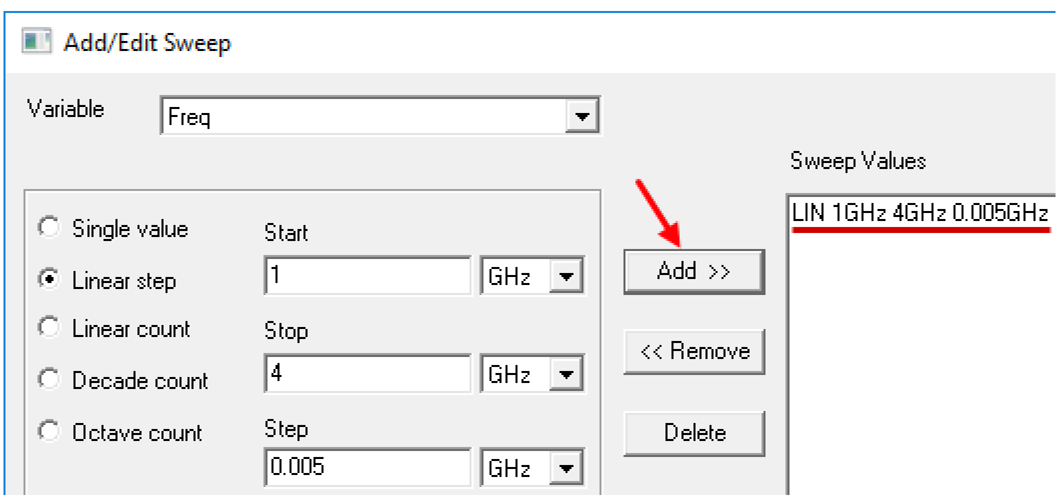
You will now set up a frequency sweep in the Circuit design spanning from 1 to 4 GHz in 0.005 GHz increments.

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)*.

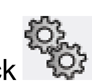
   The *Linear Network Analysis, Frequency Domain* dialog box appears.

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.

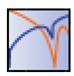
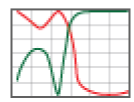
5 - Tune Matching Network

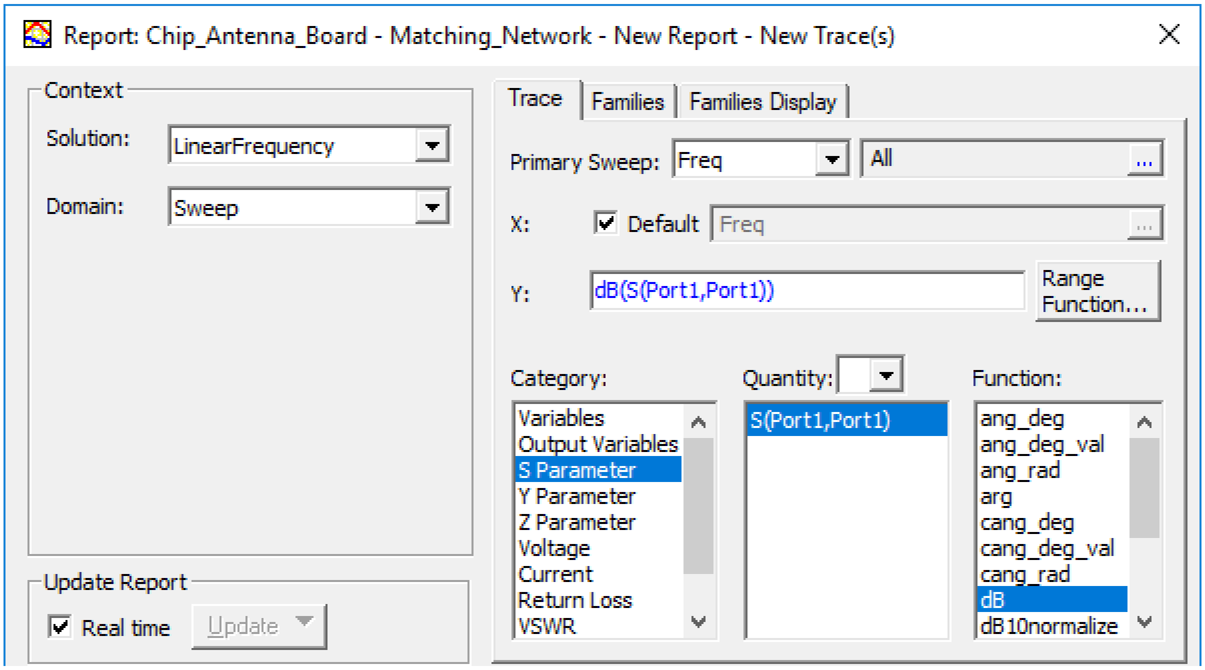
In this chapter, you will tune the component values in the matching network to minimize the signal reflection at 2.44 GHz. The goal is for the circuit and antenna to be resonant at this frequency. The process involves the following steps:

- 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

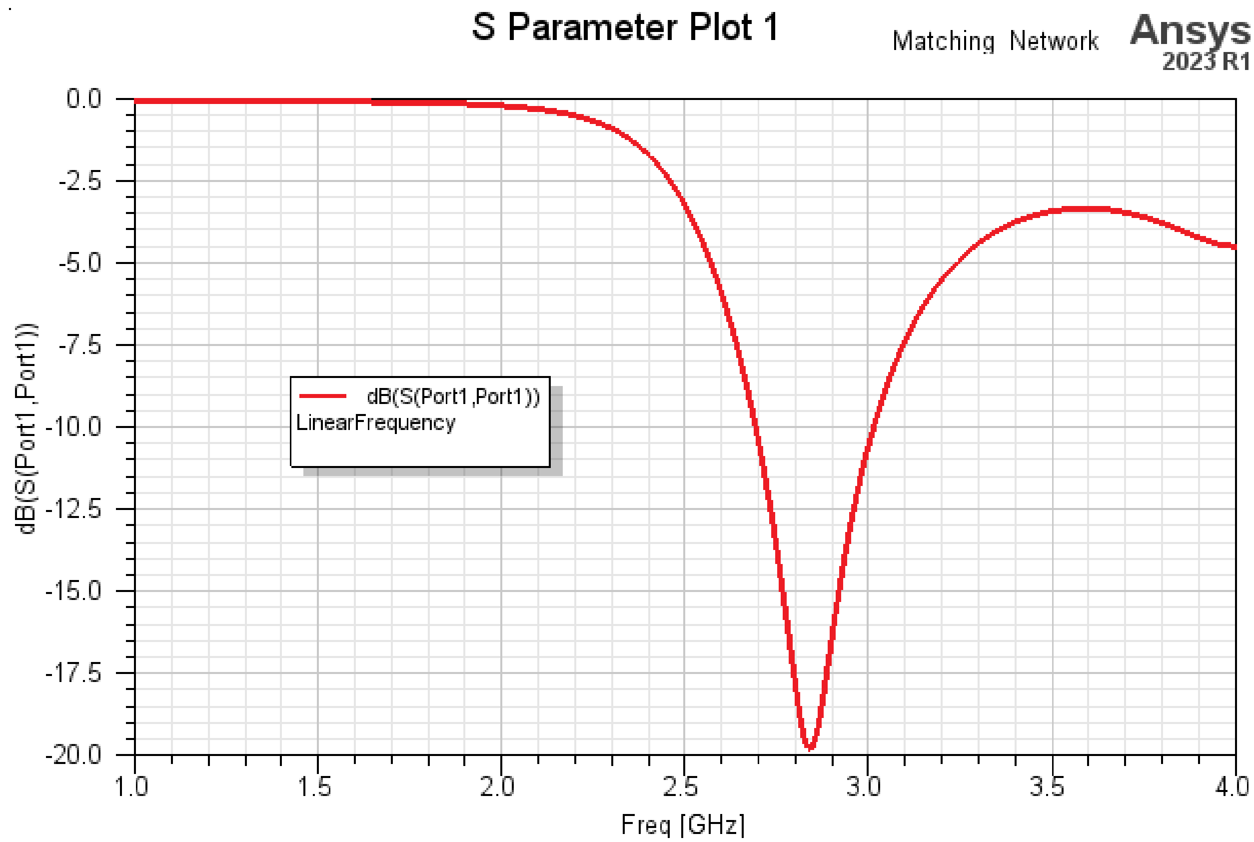
The first step in tuning the circuit is to create an S-parameter report of the signal reflection at Port1. This plot will be used as the basis of the component tunings to be performed. The signal reflection will be minimal at the resonant frequency of the circuit. Varying the capacitance and inductance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



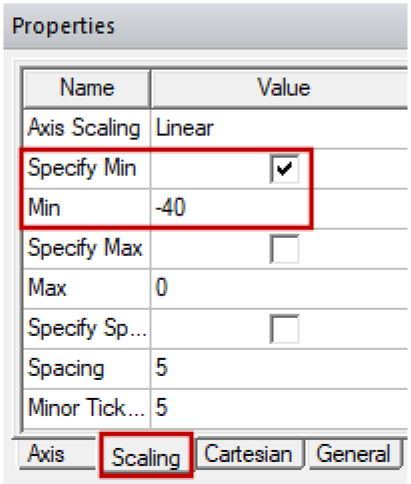
Prepare Report for Tuning

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 **-40** and press **Enter**.



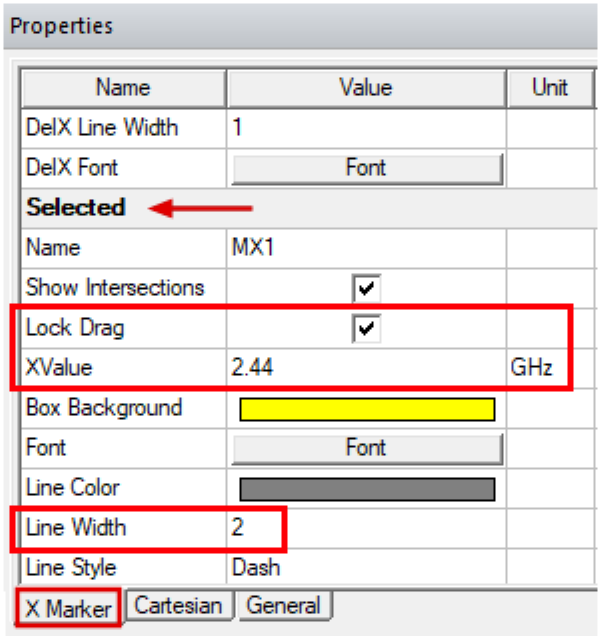
3. Right-click in the *S Parameter Plot 1* window and choose **Marker** > **Add X Marker**.

   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*).

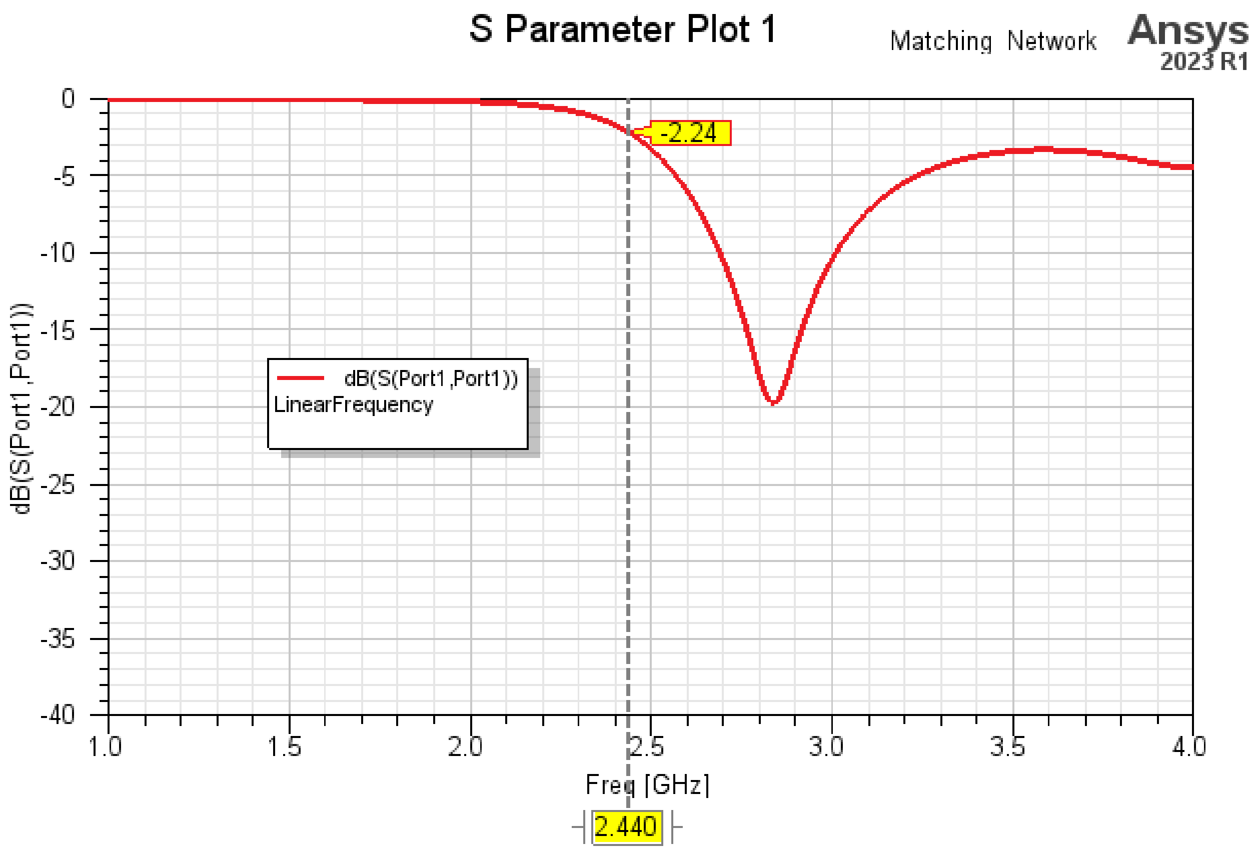
   The X marker settings appear in the docked *Properties* window.

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.

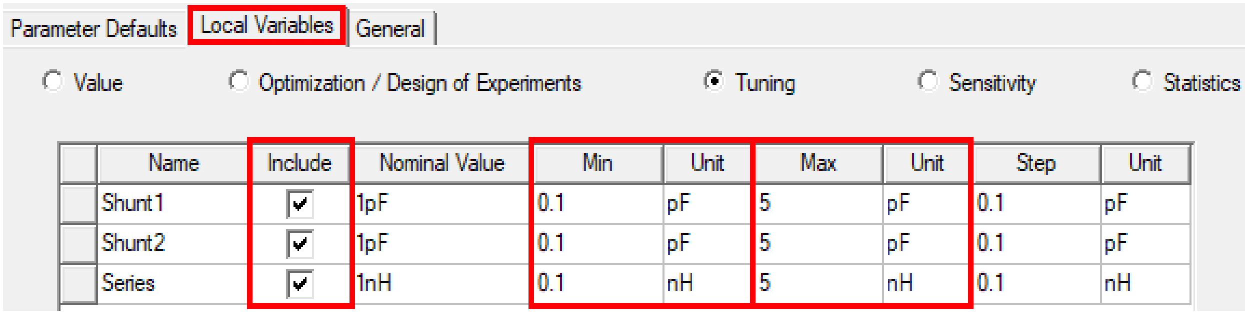
   The modified plot should look like the following image:



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: *Shunt1, Shunt2,* and *Series*.

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 check boxes in the **Include** column for all three variables (**Shunt1, Shunt2,** and **Series**).
   c. In the **Min** column, type **0.1** for all three variables. (The **Units** are **pF** for both of the **Shunt*x*** capacitors and **nH** for the **Series** inductor.)
   d. In the **Max** column, type **5** for all three variables. (The **Units** are the same as for the **Min** values.)



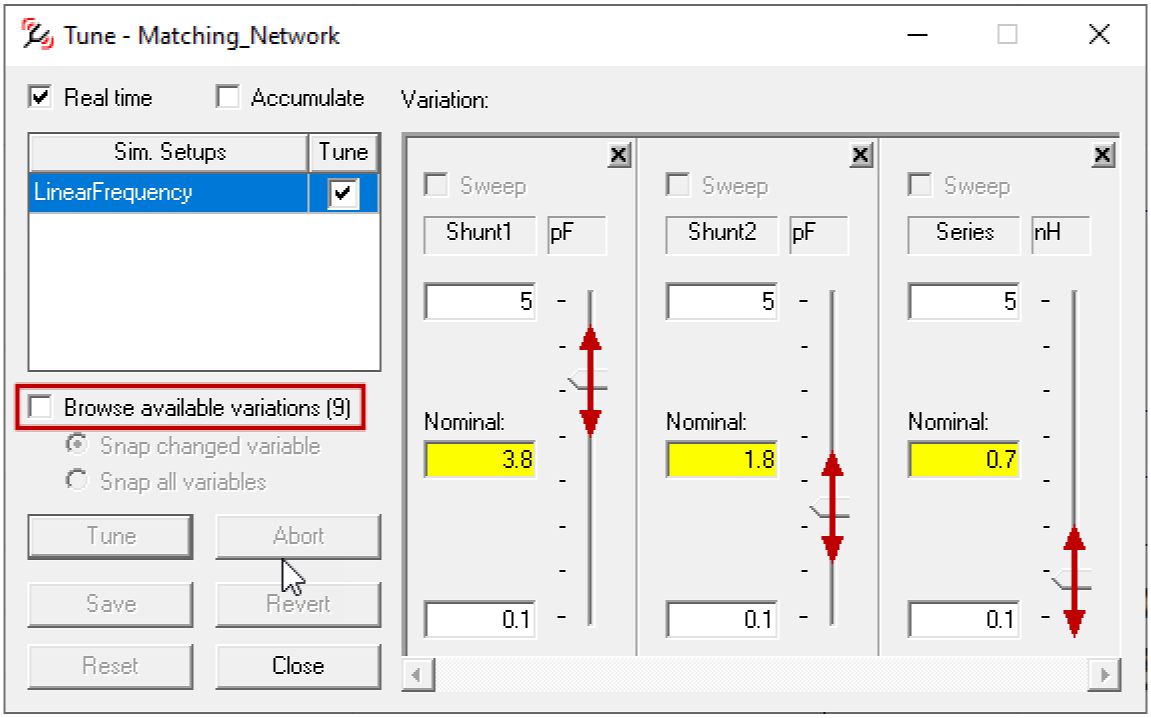
3. Click **OK** to accept the settings and close the *Properties* dialog box.

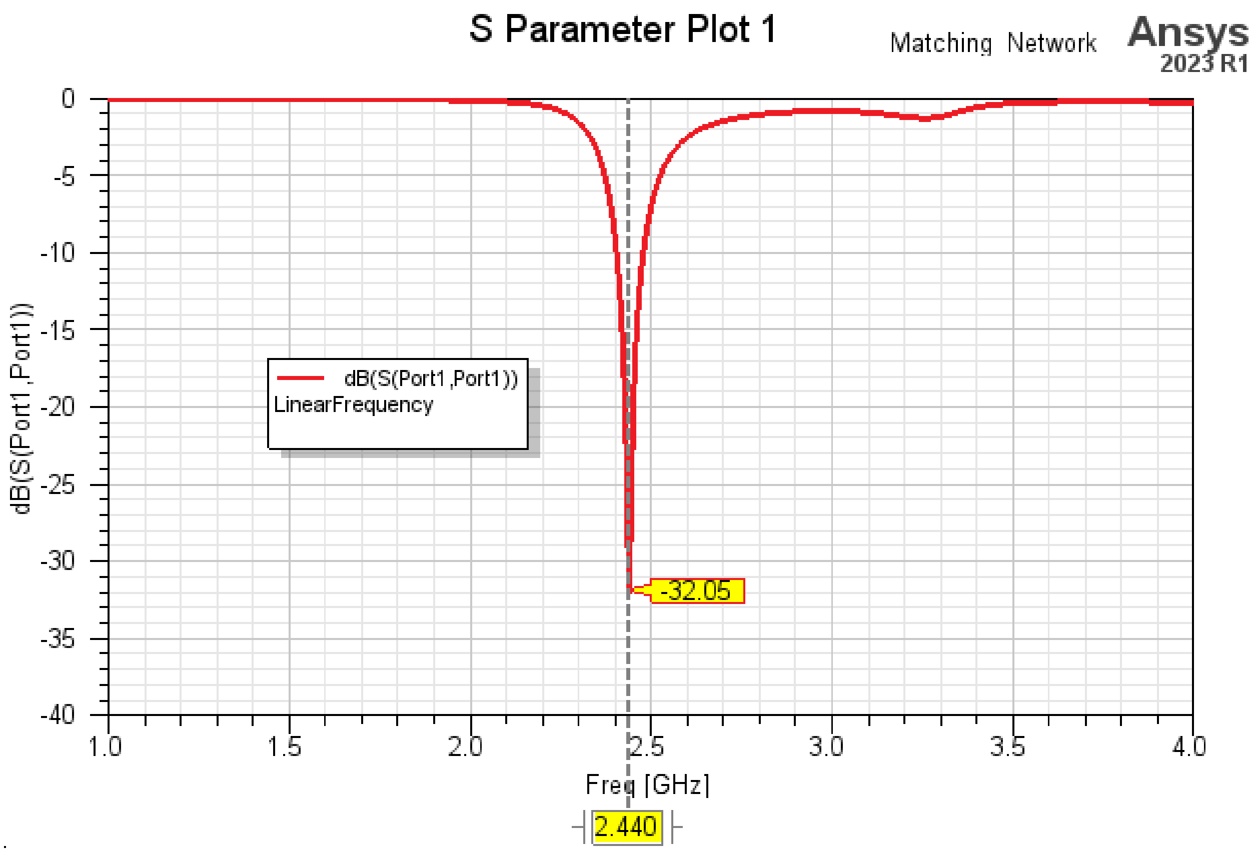
Tune Component Values

In this procedure, you will determine the *Shunt1, Shunt2,* and *Series* 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 2.44 GHz, but your decibel value may be slightly different.

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.

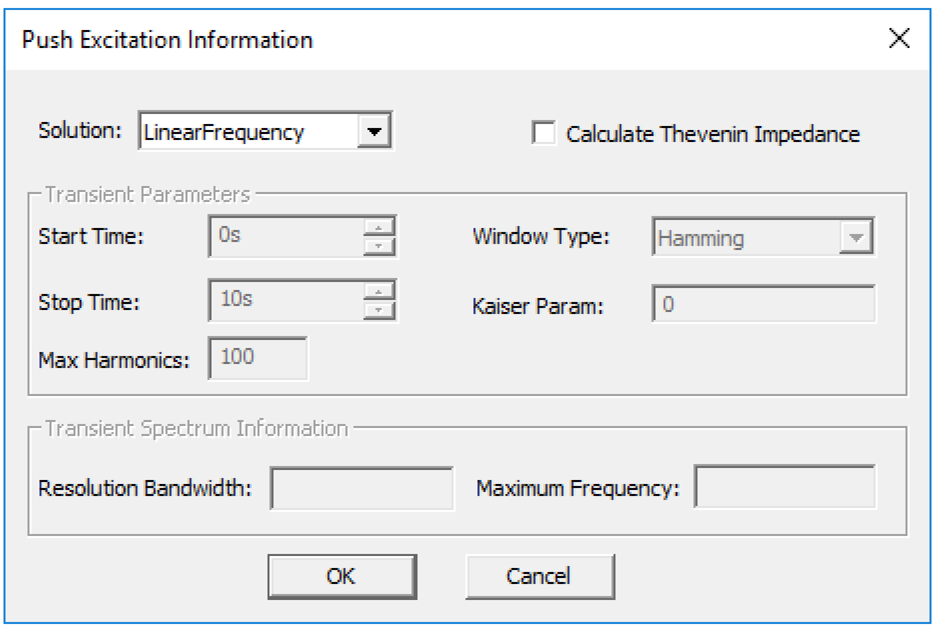
8.  **Save** the project.

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

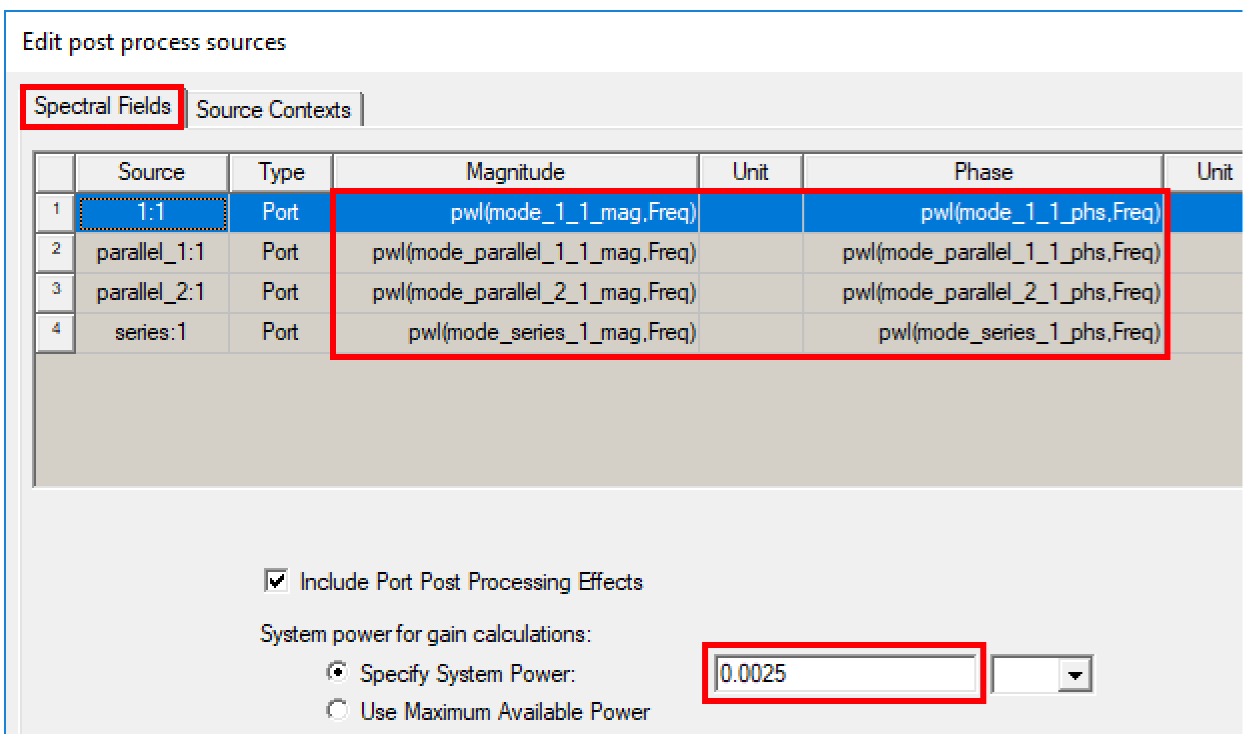
Next, verify that the excitation information has been transferred from the Circuit design to the HFSS design. This information includes the following items:

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

1. In the Project Manager, double-click **HFSSDesign*x* (DrivenModal)** to make it the active design and to bring the Modeler window to the foreground.
2. Under *HFSSDesignx (DrivenModal)* in the Project Manager, right-click **Excitations** and choose **Edit Sources** from the shortcut menu.

   The *Edit post process sources* dialog box appears.

3. 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

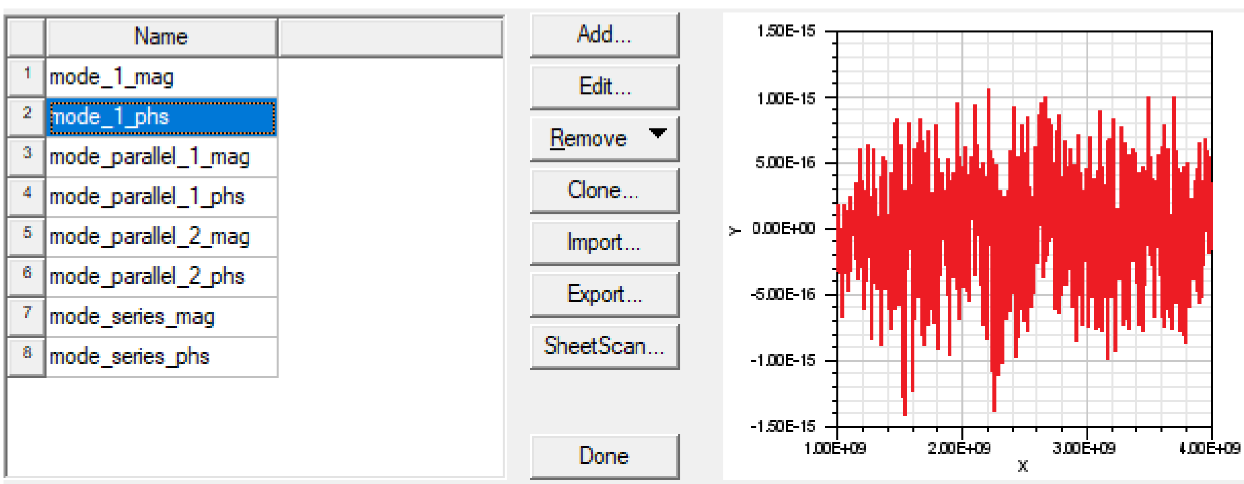
Also, notice that the *Magnitude* and *Phase* columns contain references to frequency-dependent magnitude and phase datasets. Prior to pushing the excitation information from the Circuit design, this dialog box would have shown 1 W magnitude at 0° phase for the first port and all zeros for the magnitude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

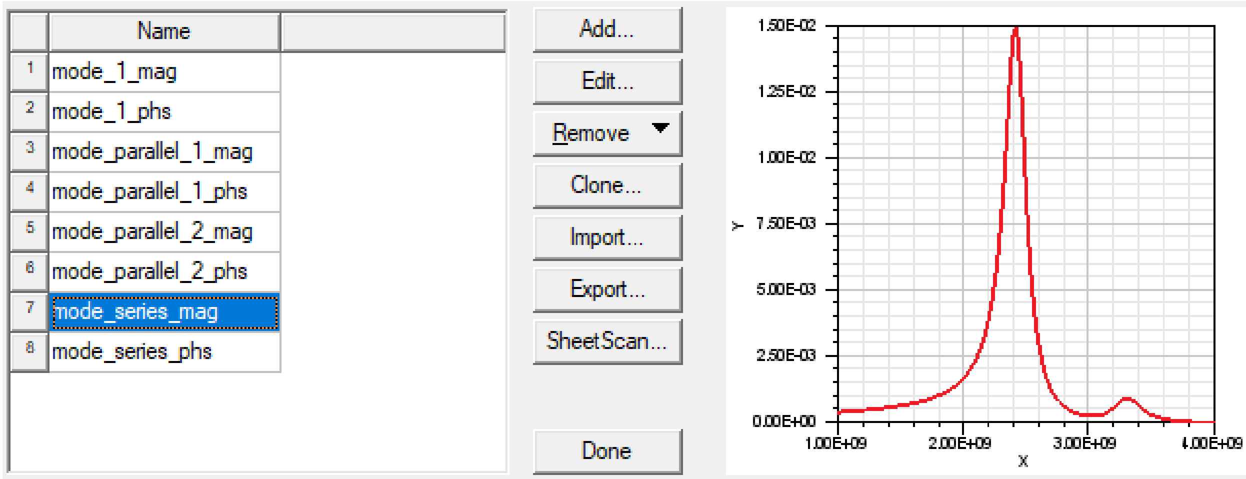
4. Click **OK**.
5. Using the menu bar, click **HFSS** > **Design Datasets**.

   The *Datasets* dialog box appears.

6. 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:

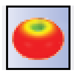
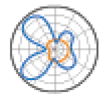


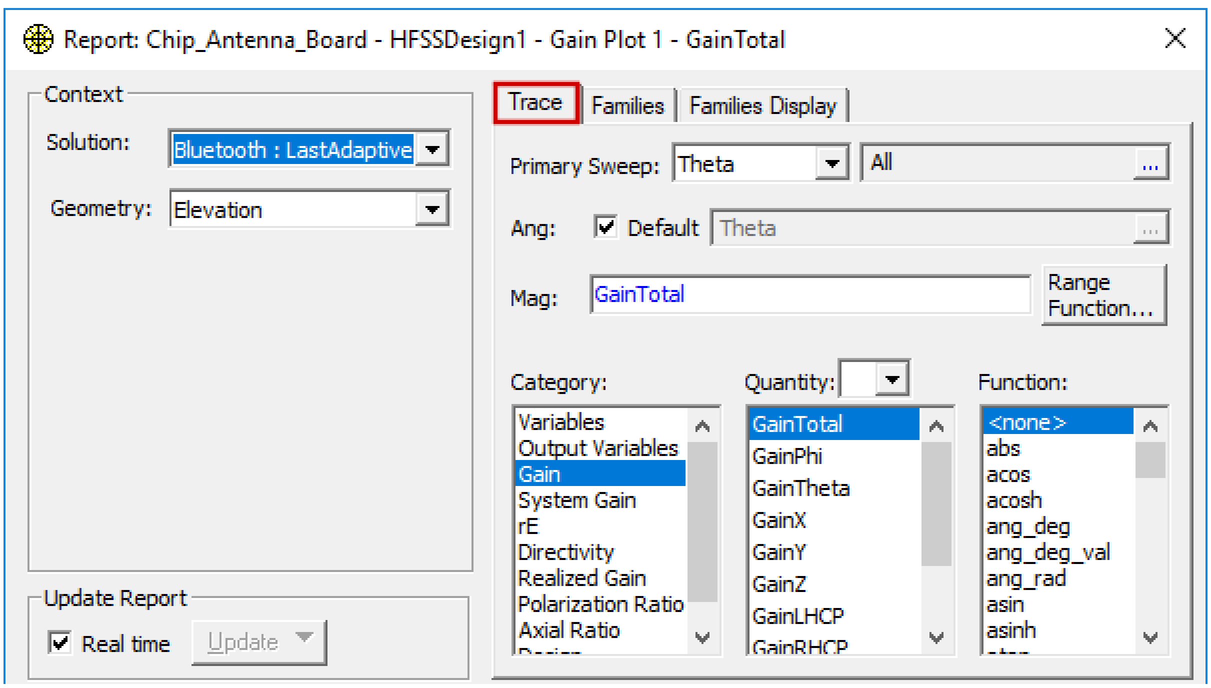

Name
1 mode_1_mag
2 mode_1_phs
3 mode_parallel_1_mag
4 mode_parallel_1_phs
5 mode_parallel_2_mag
6 mode_parallel_2_phs
7 mode_series_mag
8 mode_series_phs
Add...
Edit...
Remove
Clone...
Import...
Export...
SheetScan...
Done
1.50E-02
1.25E-02
1.00E-02
7.50E-03
5.00E-03
2.50E-03
0.00E+00
Y
1.00E+09
2.00E+09
3.00E+09
4.00E+09
X

7. Click **Done**.

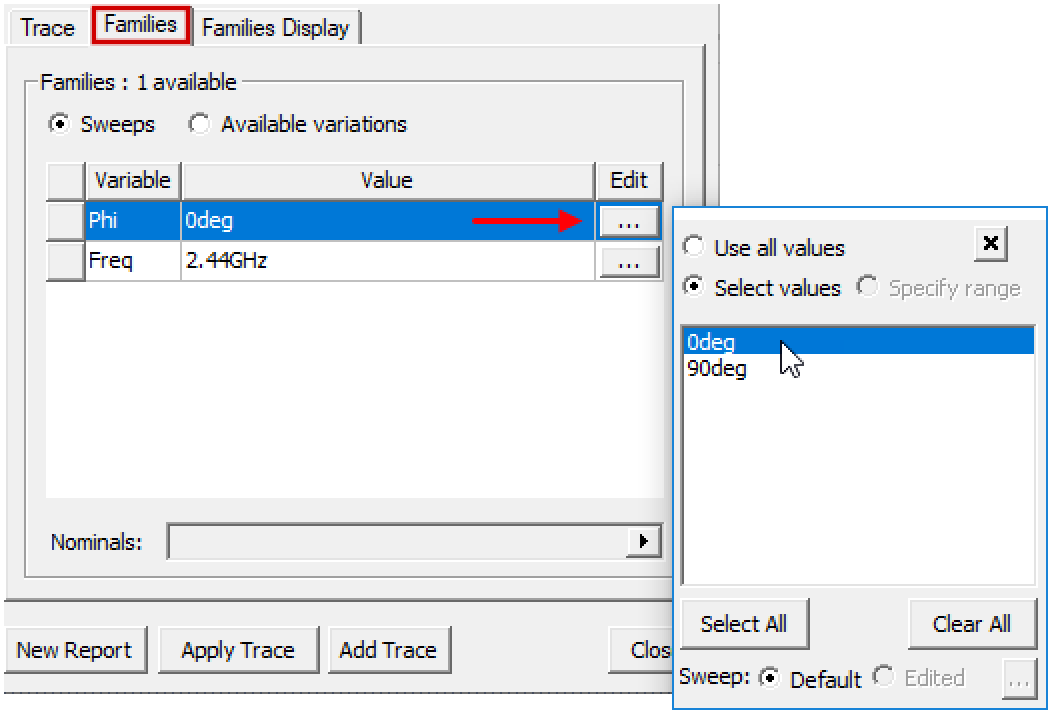
8 - Create and Overlay Gain Plots

Finally, create two total gain plots, one for Phi = 0° and one for Phi = 90°. A gain plot is a far fields report 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 plot cannot contain multiple Phi values. However, you can overlay multiple individual plots, each with single Phi values.

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:

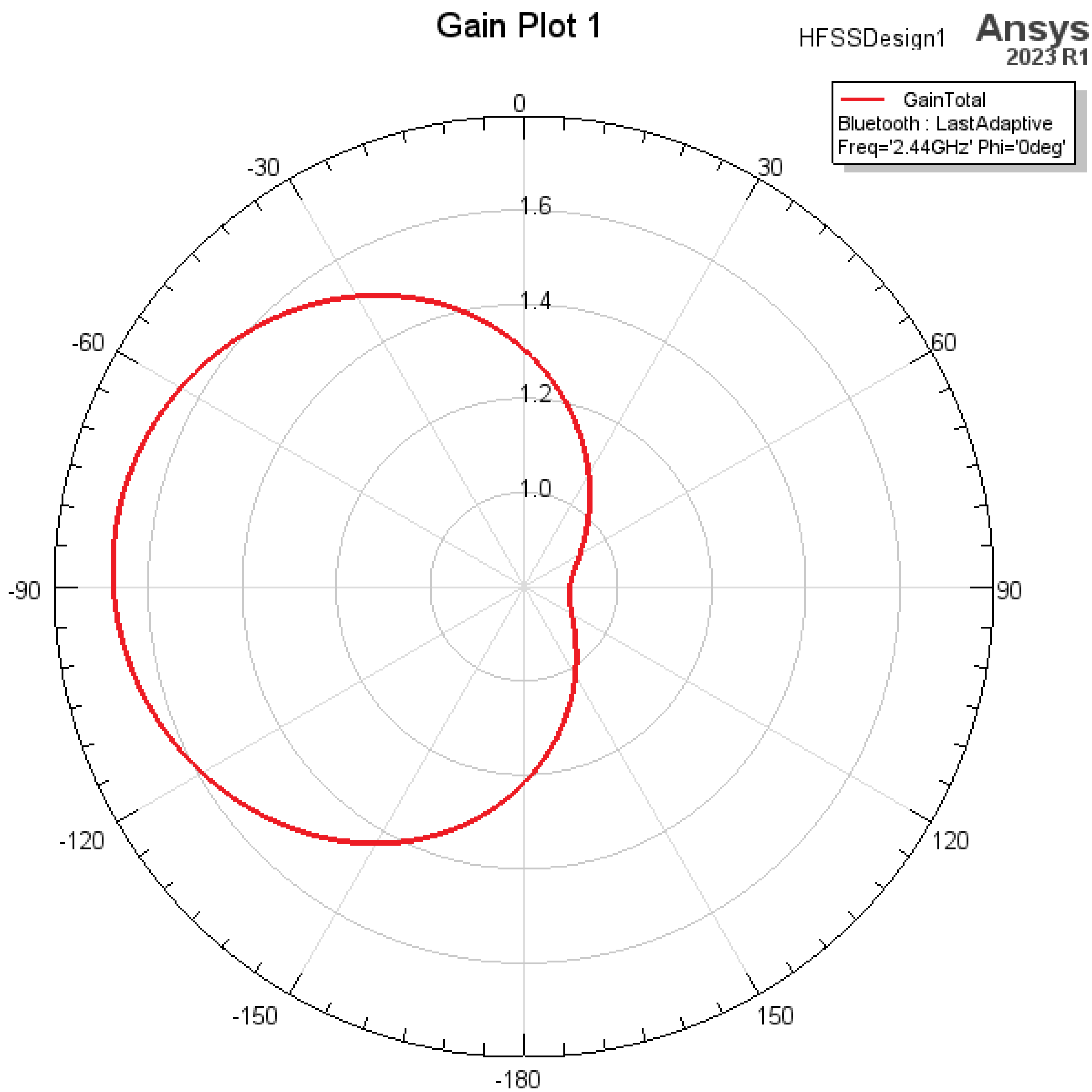


3. Under the **Families** tab, select the **Phi** value of **0deg**:



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

   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**.

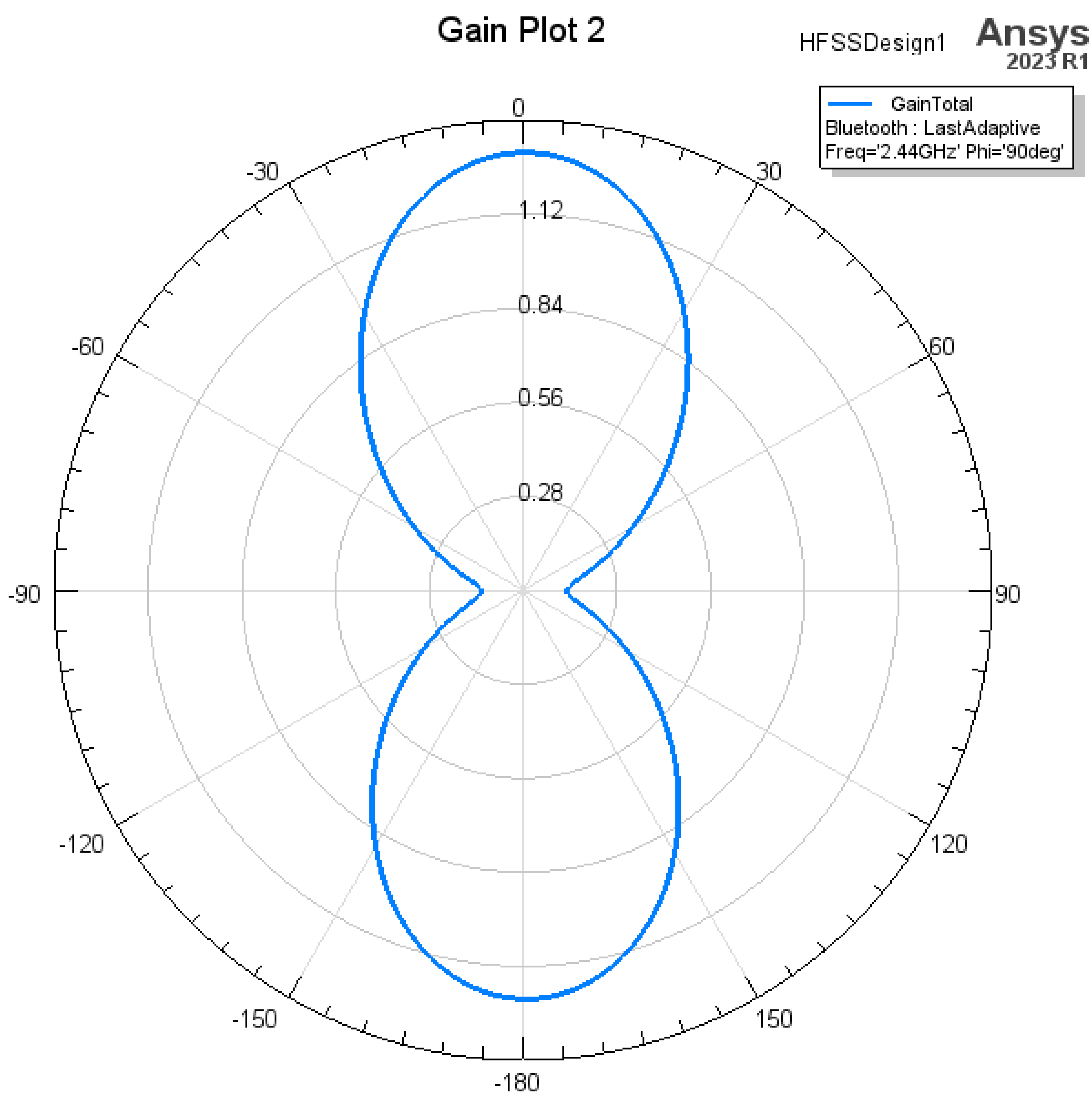
   Gain Plot 2 appears in a new window.

   To differentiate the appearance of the two gain plot curves, you will next change the color of the trace in *Gain Plot 2*.

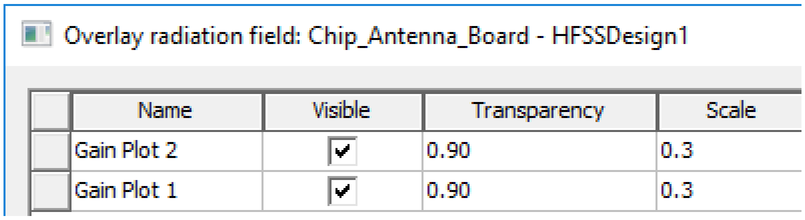
7. Click the trace in *Gain Plot 2* to select it.

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
Gain Plot 2	✔	0.90	0.3
Gain Plot 1	✔	0.90	0.3

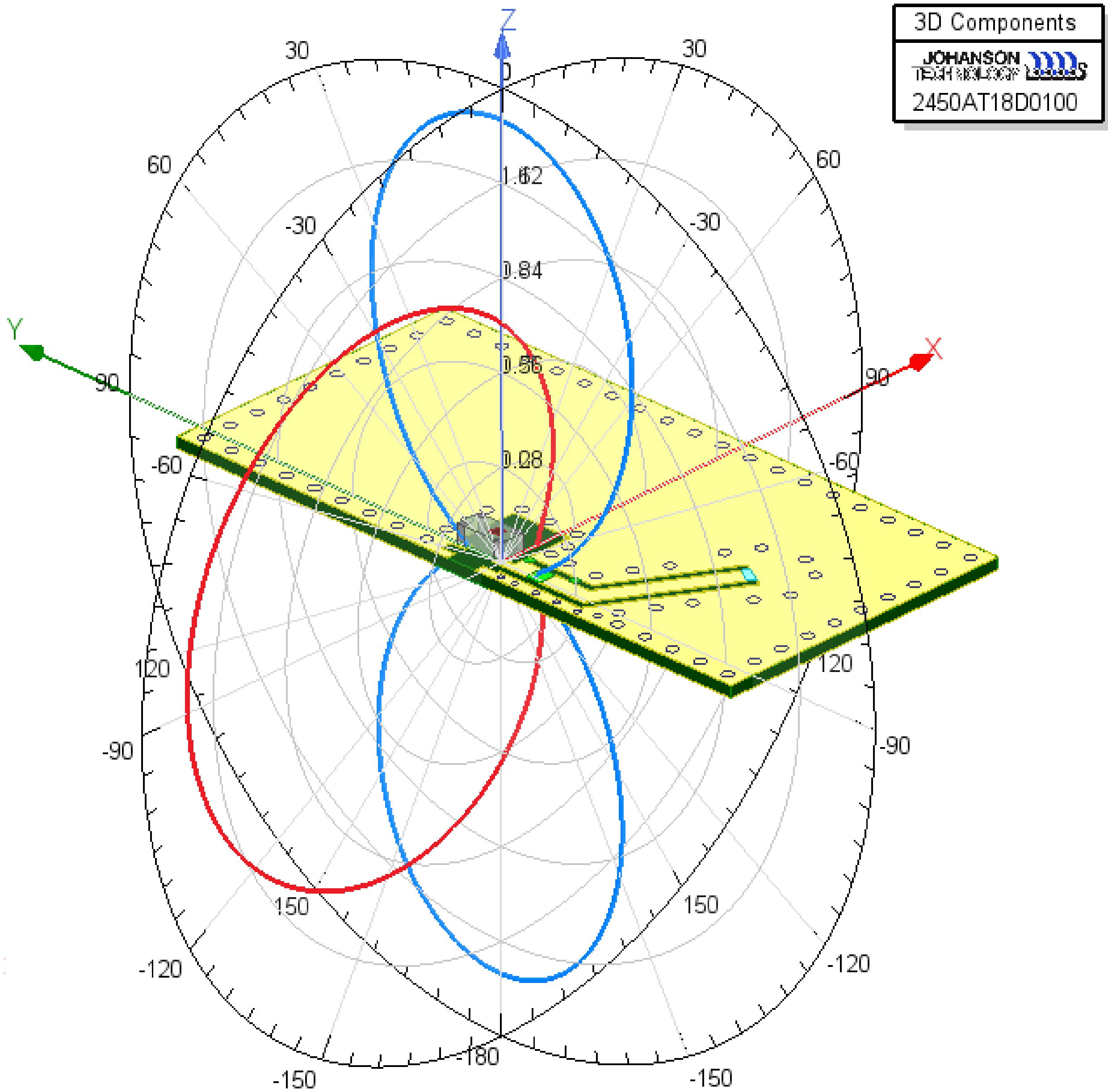
12. Click **Close**.
13. The open region surrounding the model was initially hidden in the project file you used to start the exercise. If, for whatever reason, the open region is now visible, click it (in *Object* selection mode), or select it from the History Tree (*Model* > *Solids* > *vacuum* > **RadiatingSurface**). Then, on the **Draw** ribbon tab, click **Hide selected objects in active view**.

    Next, rotate the model viewpoint for the best view of the two overlays, as follows:

14. On the **Draw** ribbon tab, click **Orient** > **Back**.
15. Hold down the middle mouse button and drag the mouse slowly toward the left and downward to produce an isometric view looking at the model from the back corner.

> **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:



16. **Save** the project.

9 - Optionally, Restore Current View Orientations

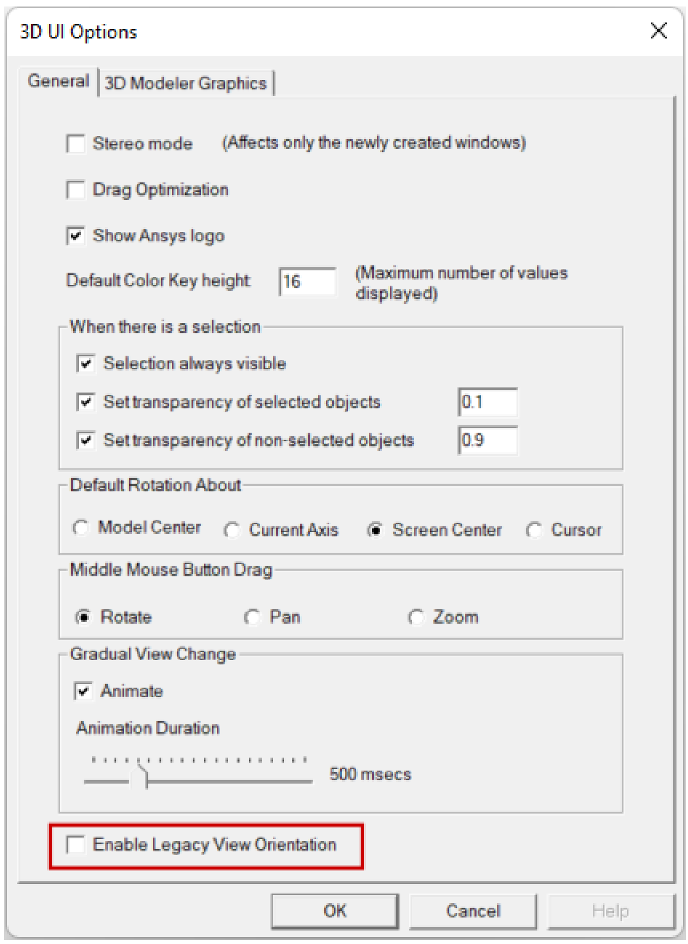
You have completed this getting started guide.

If you prefer to use the new view orientations implemented in version 2024 R1 of the Ansys Electronics Desktop application, clear the *Use Legacy View Orientation* option as follows:

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**.

The settings in the 3D UI Options dialog box are global. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

You can now save and close this project.