Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

® 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

# Getting Started with HFSS™: Silicon Spiral Inductor

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.

**Copyright and Trademark Information**

© 1986-2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

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

**Disclaimer Notice**

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

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

**U.S. Government Rights**

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

**Third-Party Software**

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



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

### 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** > **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**"

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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** > **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.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Table of Contents

**Table of Contents** ..... **Contents-1**
**1 - Introduction** ..... **1-1**
Sample Project – Silicon Spiral Inductor ..... 1-1
**2 - Set Up the Project** ..... **2-1**
Launch Ansys Electronics Desktop ..... 2-1
Set General Options ..... 2-2
Insert HFSS Design ..... 2-3
Enable Legacy View Orientations ..... 2-5
Set Model Units ..... 2-7
Verify Solution Type ..... 2-7
**3 - Construct the Model** ..... **3-1**
Create Dielectric Objects ..... 3-1
Create Substrate ..... 3-1
Create Oxide ..... 3-4
Create Passivation ..... 3-7
Create Air Body ..... 3-10
Create Conductors ..... 3-11
Create Ground Plane ..... 3-12
Hide All Existing Objects ..... 3-13
Create Spiral Inductor ..... 3-14
Define Conductor Material ..... 3-14
Define an Offset Coordinate System ..... 3-15
Create Spiral Path ..... 3-16
Assign Width and Thickness to Spiral ..... 3-19
Create Underpass ..... 3-20
Create Vias ..... 3-22
Create Feed ..... 3-26
Unite Spiral Objects ..... 3-28


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Ground Ring ... 3-30
Create a Second Offset CS ... 3-31
Define Ground Ring Material ... 3-32
Create Outer Box ... 3-32
Create Inner Box ... 3-34
Complete the Ring ... 3-36
Create Extensions ... 3-38
Unite Ring Objects ... 3-40
**4 - Assign Boundaries and Excitations ... 4-1**
Create Signal Sources ... 4-1
Assign Excitation at Sources ... 4-4
Assign Radiation Boundary ... 4-7
Assign Perfect E Boundary to Ground ... 4-9
Boundary Display (Optional) ... 4-10
**5 - Analyze the Spiral Inductor ... 5-1**
Create Analysis Setup ... 5-1
Add a Frequency Sweep ... 5-3
Assign Mesh Refinement ... 5-5
Validate and Analyze ... 5-6
Review Solution Data ... 5-7
Review the Profile Panel ... 5-8
Review the Convergence Panel ... 5-9
Review the Matrix Data Panel ... 5-11
Review the Mesh Statistics Panel ... 5-14
Create S-Parameter vs. Frequency Plot ... 5-14
Custom Equations – Output Variables ... 5-16
Simulate with Solve Inside Conductors ... 5-25
Results with Solve Inside ... 5-26
Direct Comparison of Results ... 5-32
**6 - Optionally, Restore Current View Orientations ... 6-1**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 1 - Introduction

This document is intended as supplementary material to HFSS for beginners and advanced users. It includes instructions to create, simulate, and analyze a silicon spiral inductor model.

This chapter contains the following topic:

- Sample Project - Silicon Spiral Inductor

## Sample Project – Silicon Spiral Inductor

In this project, we will use HFSS to create, analyze, and review the results of a 2.5 turn spiral inductor.

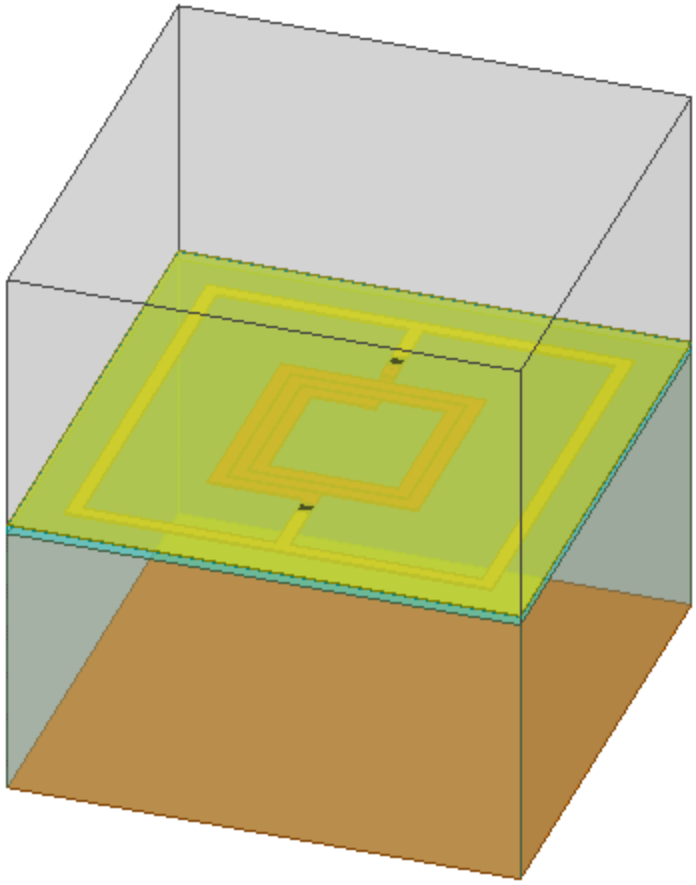


**Figure 1-1: Spiral Inductor**

This nominal design consists of the following components with their corresponding dimensions and material properties:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Dielectric Layers:** For all, Depth (XSize) = 540 µm, Width (YSize) = 540 µm
  - **Passivation:** Thickness (ZSize) = 0.7 µm, $\varepsilon_r$ = 7.9
  - **Oxide:** Thickness = 9.8um, $\varepsilon_r$ = 4.0
  - **Substrate:** Thickness = 300 µm, $\varepsilon_r$ = 11.9, σ = 10 S/m

  where $\varepsilon_r$ is the relative permittivity, and σ is the bulk conductivity of the material.

- **Conductors:**
  - **Spiral (M6):** Thickness = 2 µm, Trace Width =15 µm, Trace Spacing =1.5 µm, Inside Radius =60 µm, σ =2.8e7 S/m
  - **Underpass (M5):** Thickness = 0.5 µm, Trace Width =15 µm, σ = 2.8e7 S/m

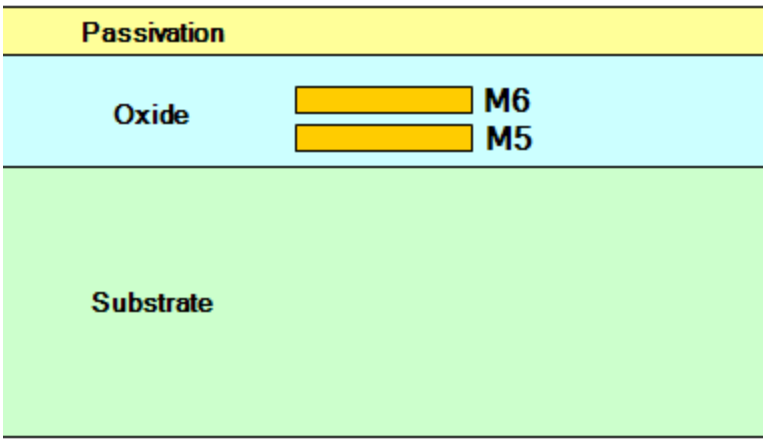



**Figure 1-2: Passivation, Oxide and Substrate**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 2 - Set Up the Project

This chapter contains the following topics:

- Launch Ansys Electronics Desktop
- Set General Options
- Insert an HFSS design
- Enable Legacy View Orientations
- Set Model Units (μm)
- Set Solution Type (Terminal)

## 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

**Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu).

   The Ansys Electronics Desktop application opens:

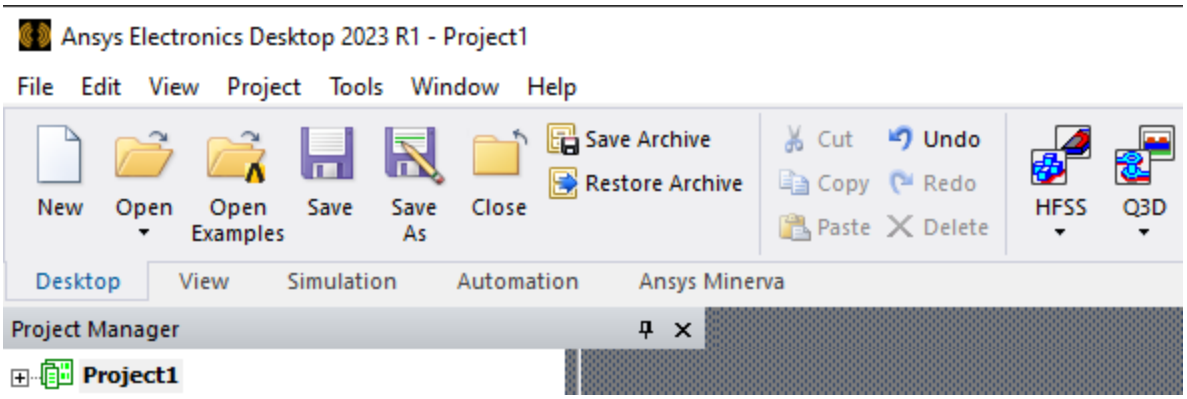



**Figure 2-1: Ansys EDT Application Launched**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If a project is not listed at the top of the Project Manager, click **New** on the **Desktop** ribbon tab to include one. If the Project Manager window does not appear after launching the application, go to the **View** menu and select the **Project Manager** option.

> **Note:**
>
> Normally, a new, project is added automatically when you launch EDT. If you had the application open already and closed the model you were working on, you will have to add a new project manually.

# Set General Options

Verify the options to be used for this exercise, as follows:

1. On the **Desktop** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. On the left side of the dialog box, expand the **HFSS** branch, select **Boundary Assignment**, and insure that all options in this group are selected:

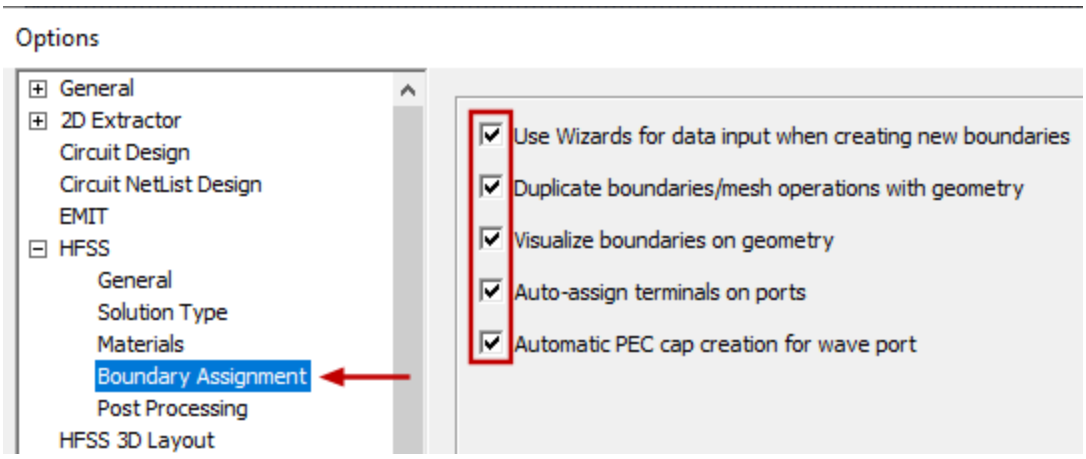



**Figure 2-2: HFSS Boundary Assignment Options**

3. Expand the **3D Modeler** branch, select **Drawing**, and ensure that the following two options are selected:
   - **Automatically cover closed polylines**
   - **Edit properties of new primitives**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

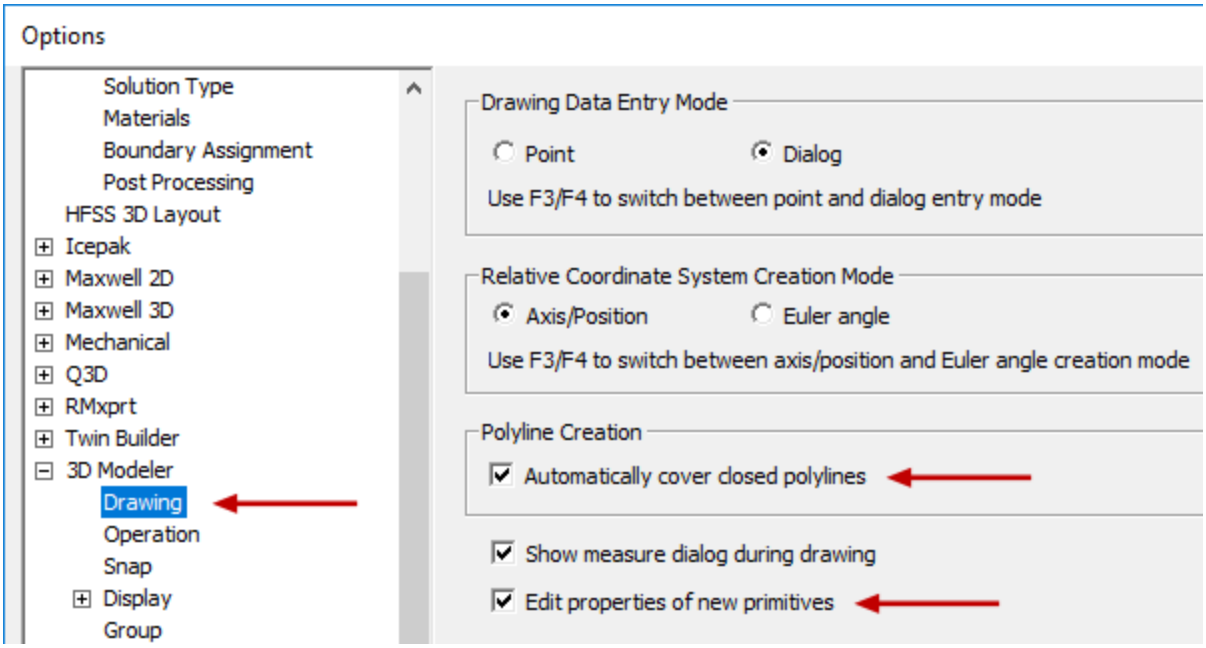



**Figure 2-3: 3D Modeler Drawing Options**

> **Note:**
>
> The *Edit properties of new primitives* option causes a *Properties* dialog box to appear automatically whenever you create a new object.

# Insert HFSS Design

Insert an HFSS design into your new project as follows:

1. On the **Desktop** ribbon tab, click **HFSS** (Insert HFSS design). (You do not have to access the HFSS drop-down menu since the default action is to insert a regular HFSS design type.)

   The *Modeler* window appears on the desktop, the ribbon advances to the *Draw* tab, and **HFSSDesign*x*** appears under **Project*x*** in the Project Manager:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

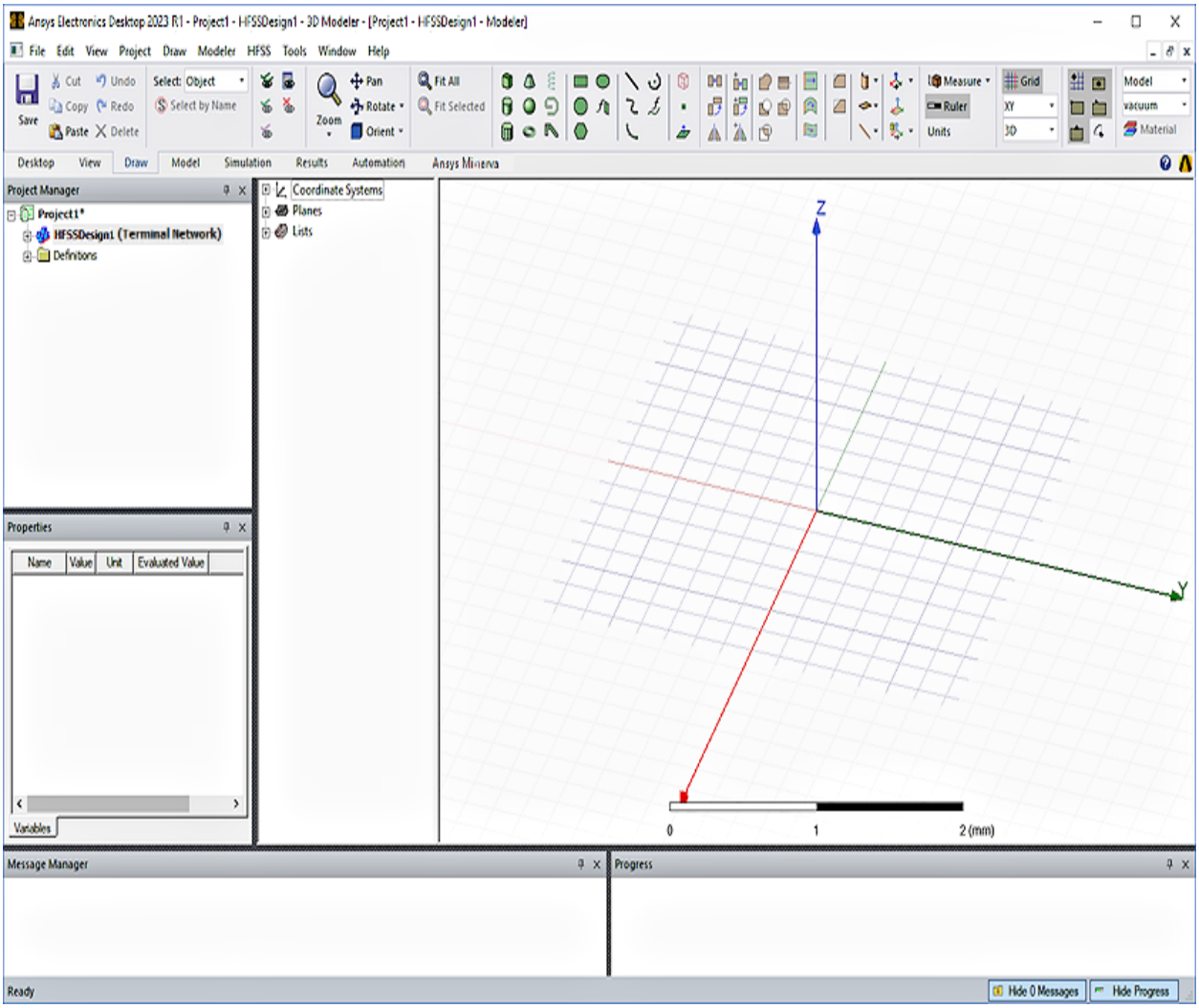



**Figure 2-4: HFSS Design Added to the Project**

**Note:**

- Adding an HFSS design modifies the project. In the Project Manager, an asterisk appears after the project name to indicate that there are unsaved changes.
- *Terminal Network* is the default solution type, unless the user has saved a different default. You will verify the appropriate solution type settings in a later step.

2. Right-click **Project*x*** at the top of the Project Manager and select **Rename** from the short-cut menu.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type **Si_Spiral_Inductor** and press **Enter**.

   The file *Si_Spiral_Inductor.aedt* is saved to your default projects folder.

### Rename the Design

You will solve two variations of the spiral inductor design. Rename this design to differentiate it from the later variant, as follows:

4. Right-click **HFSSDesign1 (Terminal Network)** in the Project Manager and choose **Rename**, change the name to **No_Solve_Inside**, and press **Enter**.

# Enable Legacy View Orientations

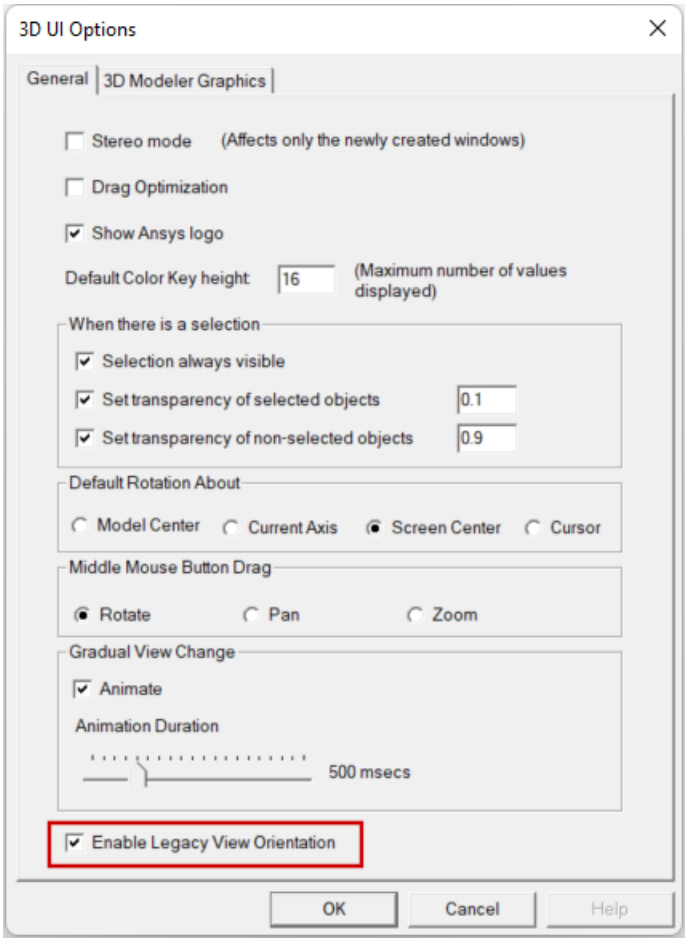
This getting started guide was created based on standard view orientations that were in effect for version 2023 R2 and earlier of the Ansys Electronics Desktop application. For consistency between your experience and the views and instructions contained in this guide, select the *Enable Legacy View Orientation*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


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.




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.

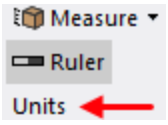
For a comparison of the legacy and current view orientations, search for *"View Options: 3D UI Options"* in the HFSS help. Additionally, views associated with **Alt + double-click** zones have


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

been redefined. The current orientations are shown in the help topic, *"Changing the Model View with Alt+Double-Click Areas."*

# Set Model Units

Set the length unit for the geometric model as follows:

1. On the **Draw** ribbon tab, click **Units**. ( 

There is no icon associated with this command.)

   The *Set Model Units and Max Extent* dialog box appears.

2. Choose **um** (micron or $10^{-6}$ meter) from the **Select units** drop-down menu.

   Keep the **Rescale to new units** and **Advanced** options cleared.

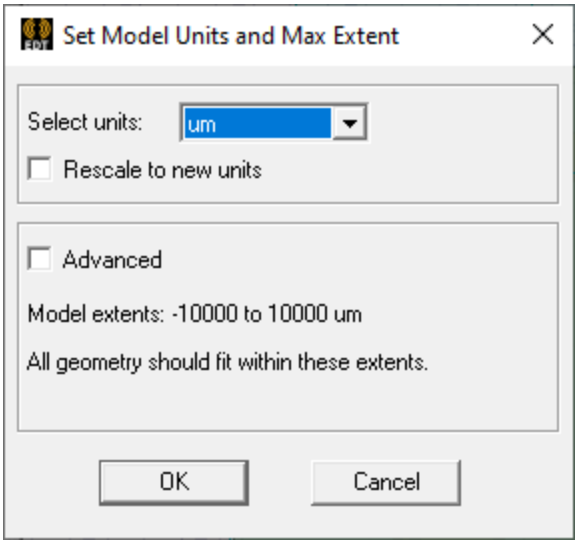



**Figure 2-5: Choosing the Length Unit**

3. Click **OK**.

# Verify Solution Type

To verify the solution type and change it if necessary:

1. Using the menu bar, click **HFSS** > **Solution Type**.

   The *Solution Type* dialog box appears.

2. Ensure that the solution type and options are set as shown in the following image:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

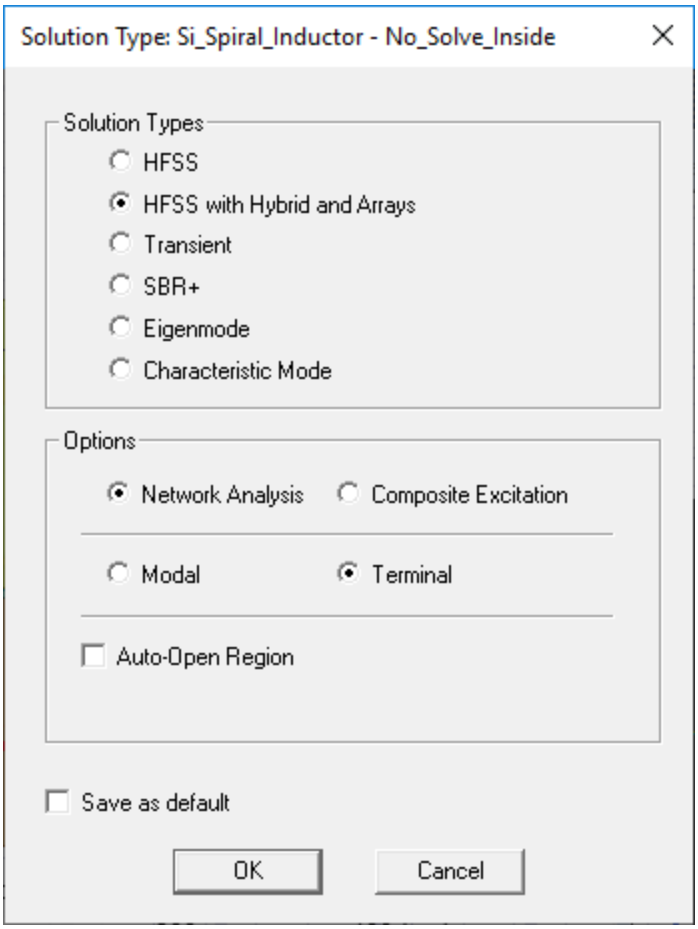



**Figure 2-6: *Solution Type* Dialog Box**

> **Note:**
>
> Terminal solutions calculate the terminal-based S-parameters of multi-conductor transmission line ports. The S-matrix solutions will be expressed in terms of terminal voltages and currents.

3. Click **OK**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 3 - Construct the Model

This chapter describes how to build the 3D spiral inductor model in HFSS (including drawing the geometric objects and assigning materials, boundaries, and excitations). The drawing operations are grouped by dielectric and conductor objects.

The following sections are covered in this chapter:

- Create Dielectric Objects
- Create Conductors
- Assign Boundaries and Excitations

During the construction process, you will hide certain objects to facilitate the drawing of other objects and then make them visible again afterward.

## Create Dielectric Objects

You will begin construction by drawing the nonconducting (dielectric) objects. This subsection contains the following topics:

- Create Substrate
- Create Oxide
- Create Passivation
- Create Air Body

## Create Substrate

To create the substrate, first draw a box freehand as follows:

1. On the **Draw** ribbon tab, click **Draw box**.
2. Press **F3** to ensure that you are in the *point* geometry entry mode.

   The cursor is accompanied by a black diamond.

3. Click at three random points inside the Modeler window to establish the base corners and height of an arbitrary box.

   After the third click, the *Properties* dialog box appears.

4. Edit the values in the **Command** tab of the *Properties* dialog box to match the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

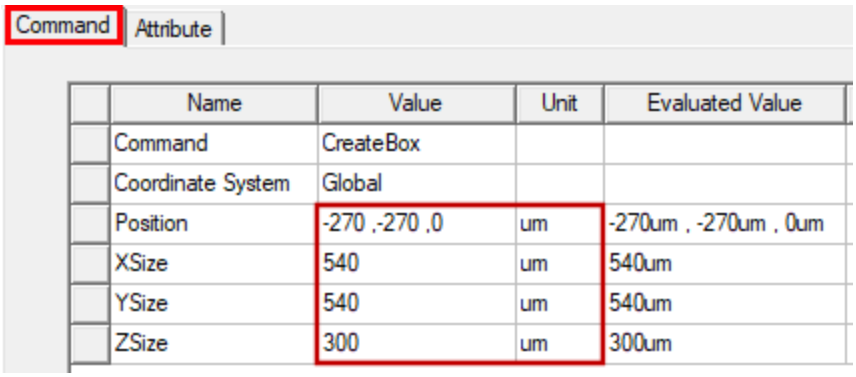



**Figure 3-1: *Substrate* Properties – *Command* Tab**

5. On the **Attribute** tab of the Properties dialog box, make the following changes:
   a. Change the **Name** to **Sub**.
   b. Select **Edit** from the **Materials** drop-down menu.

      The *Select Definition* dialog box appears.

   c. Click **Add Material**.

      The *View / Edit Material* dialog box appears.

   d. Edit the **Material Name, Relative Permittivity,** and **Bulk conductivity** values as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

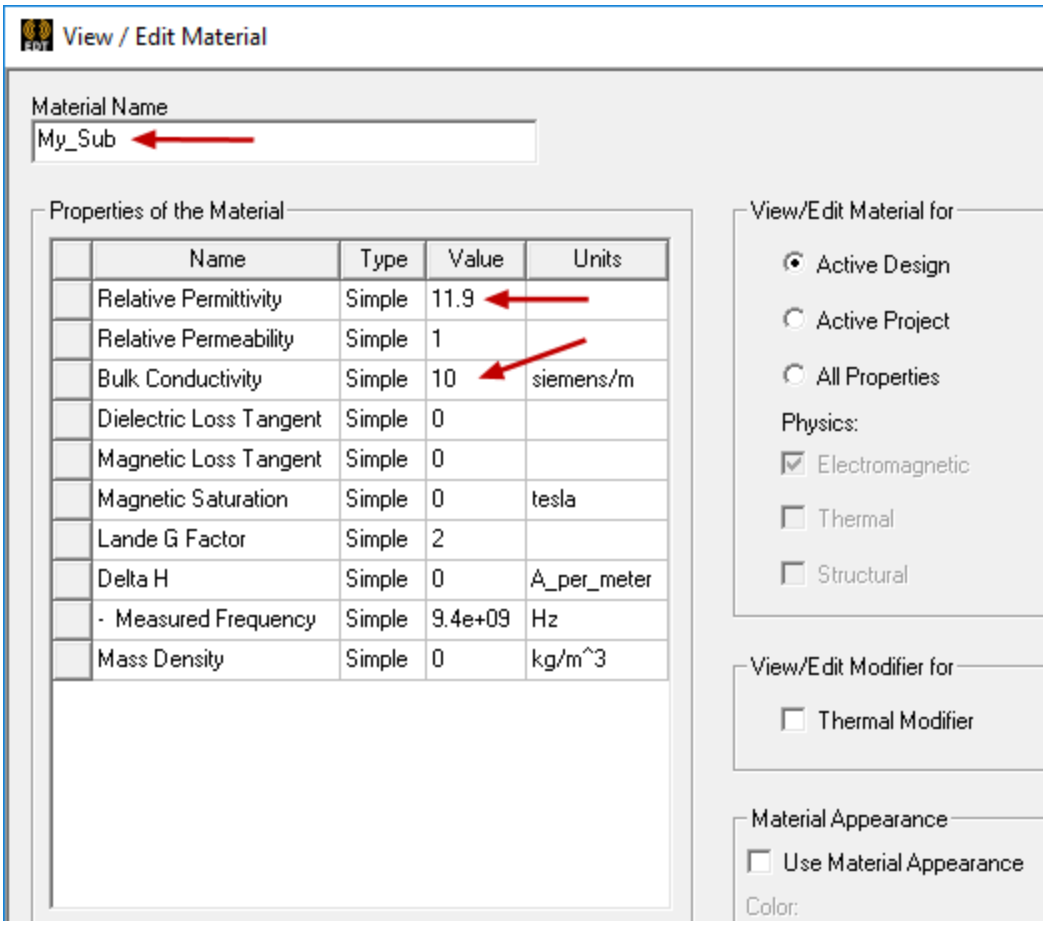
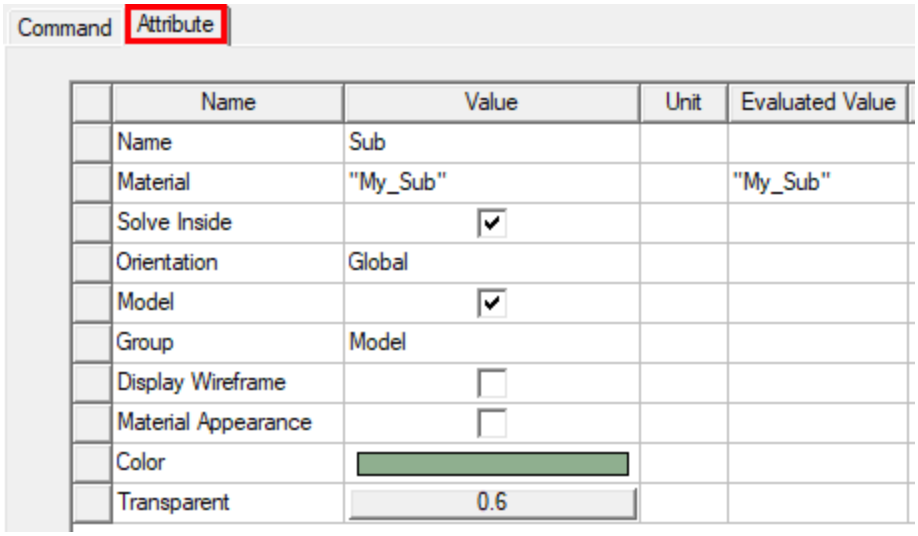



**Figure 3-2: Substrate Material Properties**

e. Click **OK** twice to close the *View/Edit Material* and *Select Definition* dialog boxes.

f. Change the **Transparent** value to **0.6**.






Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

**Figure 3-3: Substrate Attributes**

6. Click **OK** to close the *Properties* dialog box.
7. Press **Ctrl+D** to fit the model to the viewing area and click in the Modeler window's background area to clear the selection.

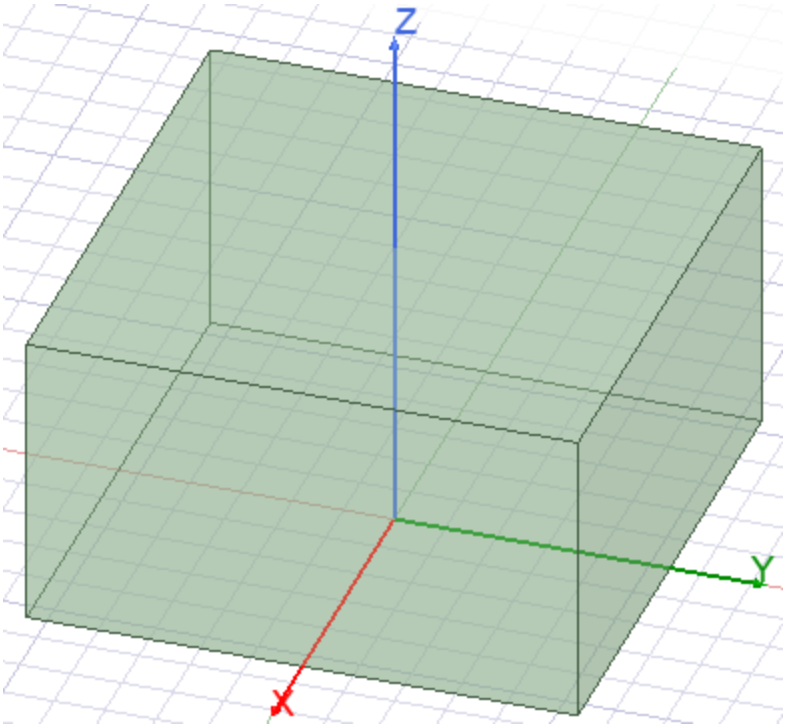



**Figure 3-4: Substrate Created**

# Create Oxide

The oxide layer sits directly on top of the substrate and has the same X and Y size dimensions. You can snap to two opposite corners of the substrate top to define the base rectangle for the oxide box. Then, click an arbitrary height point, edit the Z size, and define the name, material, and appearance properties, as follows:

1. On the **Draw** ribbon tab, click **Draw box**. Then:
    a. Click the upper left and then the lower right corners of the substrate's top face. The cursor becomes a black square to indicate the snapping point at the corner vertices.
    b. Move the mouse upward slightly and click a third time to define the box height.
2. On the **Command** tab of the *Properties* dialog box that appears, change the **ZSize** value to **9.8 µm**:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

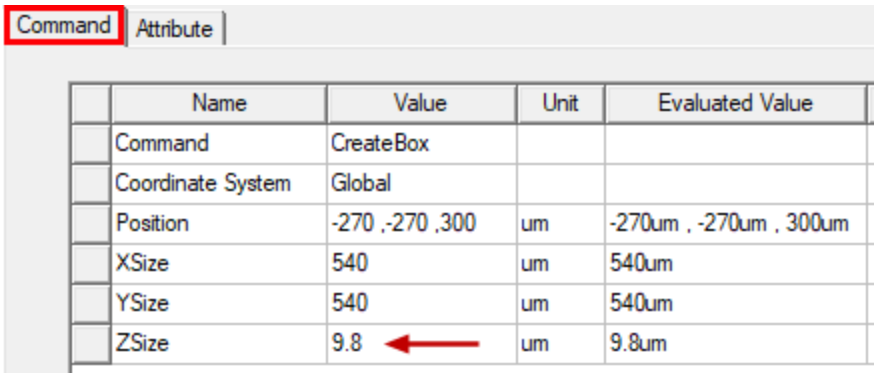



**Figure 3-5: Oxide Properties – *Command* Tab**

> **Note:**
>
> If you snapped to two different opposing corners, the sign of the *Position* coordinates, *XSize*, and/or *YSize* values may differ from the preceding figure, but the rectangle should still be correctly sized and placed.

3. On the **Attribute** tab, do the following:
   a. Change the **Name** to **Oxide**.
   b. Choose **Edit** from the **Materials** drop-down menu, click **Add Material**, and specify the properties shown below:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

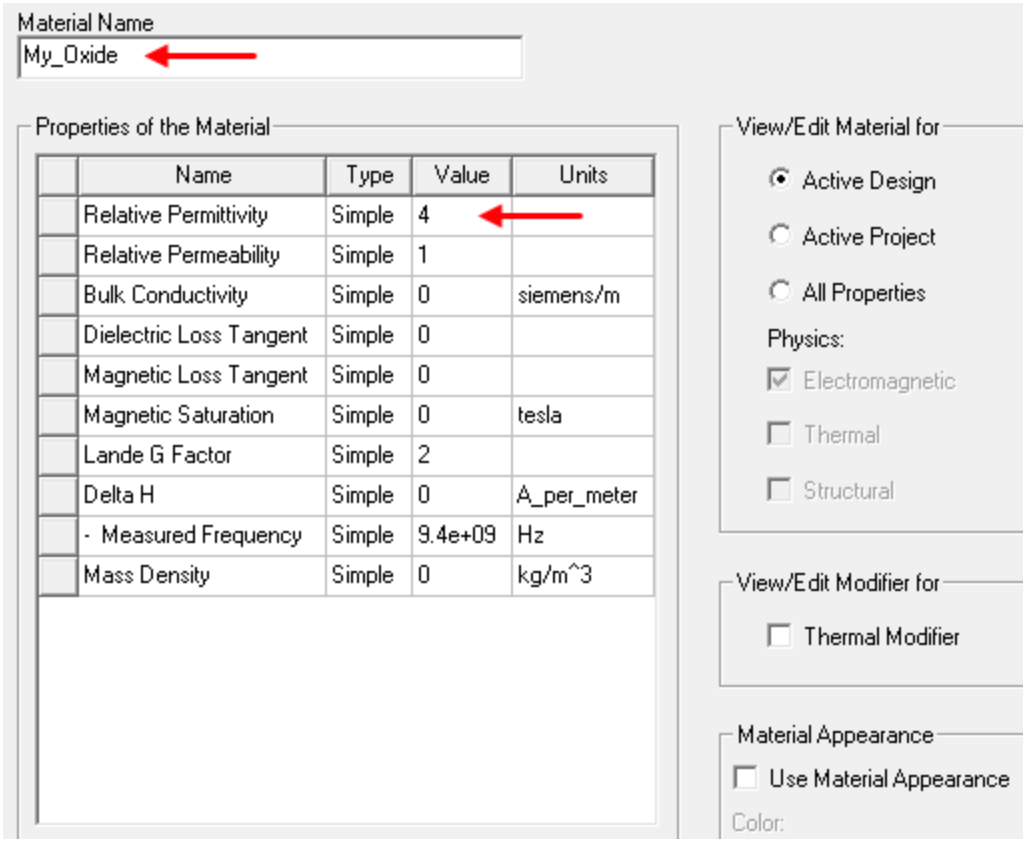
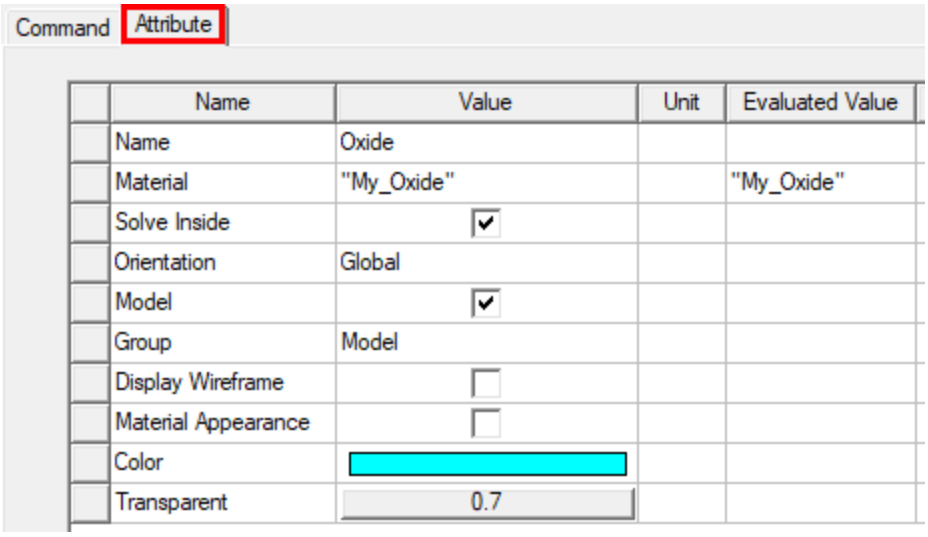



**Figure 3-6: Oxide Material Properties**

c. Click **OK** twice to close the *View/Edit Material* and *Select Definition* dialog boxes.

d. Change the **Color** to **cyan** (column 5, row 2 of the *Basic color* samples; Red: 0, Green: 255, Blue: 255).

e. Change the **Transparent** value to **0.7**.






Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

**Figure 3-7: Oxide Attributes**

4. Click **OK** to close the *Properties* dialog box.
5. Clear the selection.

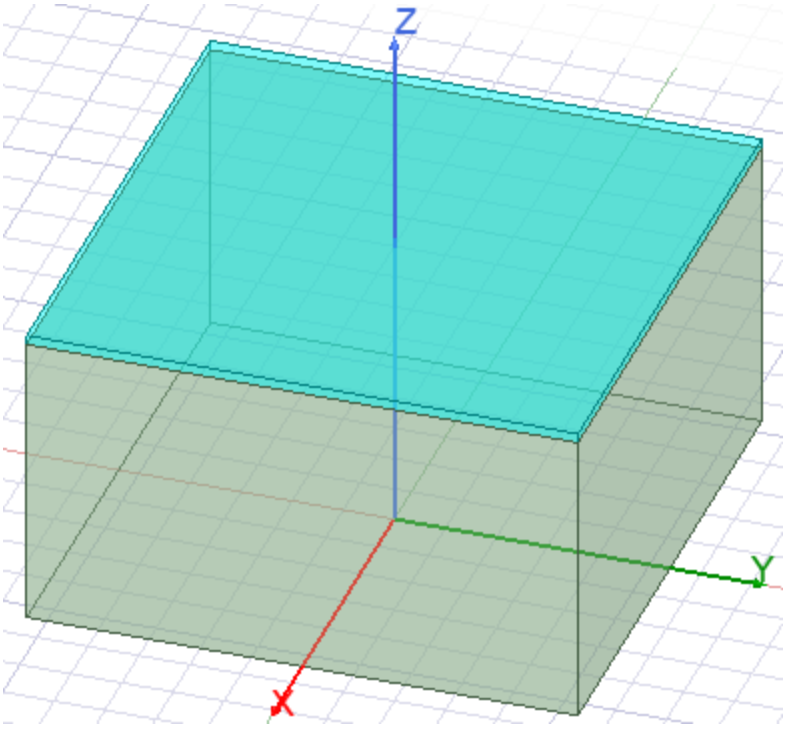



**Figure 3-8: Oxide Created**

# Create Passivation

The passivation layer sits directly atop the oxide layer and has the same X and Y size dimensions. Therefore, you can draw the box for this object using the same method as you did for the oxide layer.

1. Draw a box using the upper left and lower right corners of the oxide's top face (selected in that order) to define the base rectangle and clicking an arbitrary third point to set the height.
2. In the **Command** tab of the *Properties* dialog box that appears after the third click, change the **ZSize** value to **0.7 µm**:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

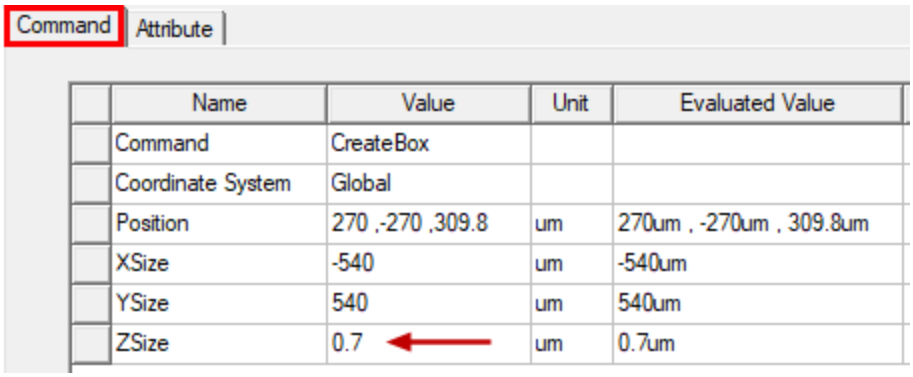



**Figure 3-9: Passivation Properties – Command Tab**

3. On the **Attribute** tab make the following changes:
   a. Change the **Name** to **Pass**.
   b. Choose **Edit** from the **Materials** drop-down menu, click **Add Material**, and specify the properties shown below:

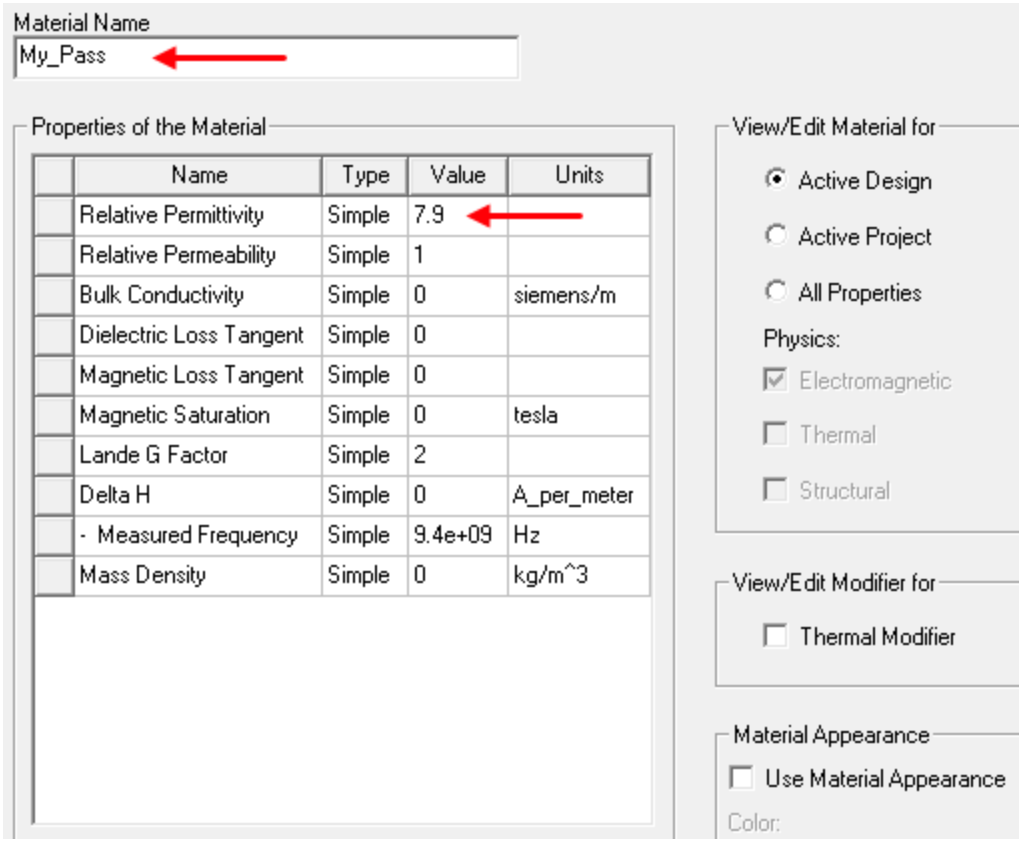



**Figure 3-10: Passivation Material Properties**

   c. Click **OK** twice to close the *View/Edit Material* and *Select Definition* dialog boxes.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Change the **Color** to **yellow** (column 2, row 2 of the *Basic color* samples; Red: 255, Green: 255, Blue: 0).

e. Set the **Transparent** value at **0.5**.

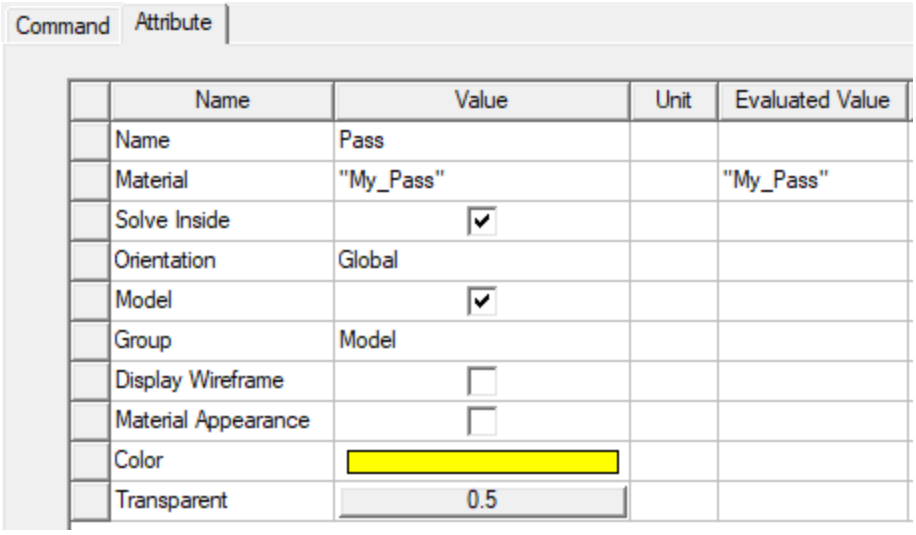



**Figure 3-11: Passivation Attributes**

4. Click **OK** to close the *Properties* dialog box.
5. Clear the selection.

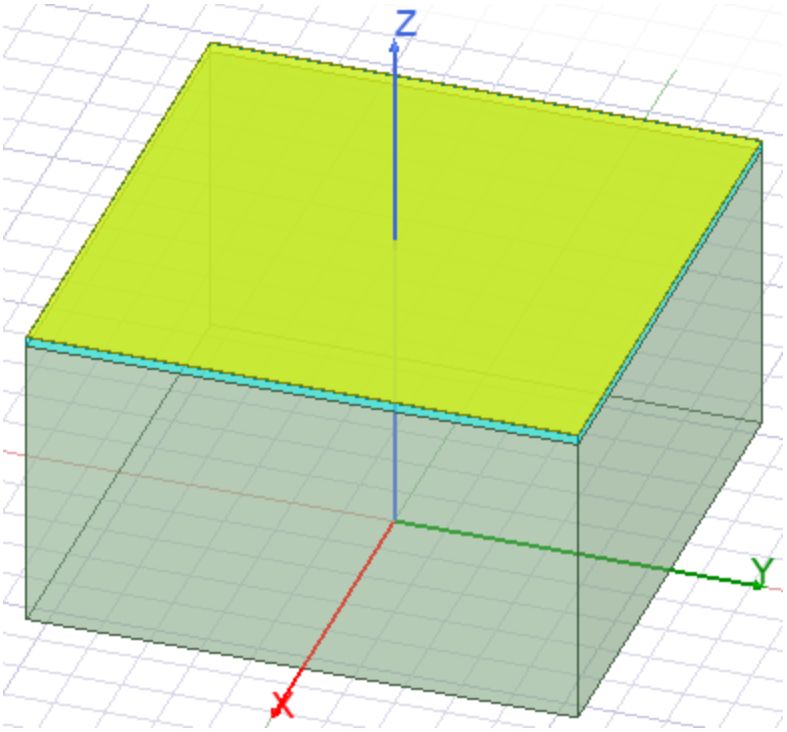



**Figure 3-12: Passivation Created**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Create Air Body

You will create a region of air around the spiral inductor and assign a radiation boundary to its outside faces. A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space. In this context, the inductor is simulated as if it's in a free space, uninfluenced by the proximity of other conductors.

The X and Y dimensions of the air body match those of the model's dielectric objects. The bottom of the air body corresponds to the bottom face of the substrate, but the air body extends to a little more than twice the substrate's height. The reason the air body is allowed to overlap the dielectric bodies is one of convenience. Radiation boundaries will be applied to the air body faces in a convenient single operation. If the air body began at the top of the passivation layer, radiation boundaries would have to be applied to five of the six air body faces and to selected faces of the substrate, oxide, and passivation layers too, which would be much less convenient.

To create an air body, draw a box and specify its size and location as follows:

1. On the **Draw** ribbon tab, click **Draw box**. Then:
    a. Click the bottom left corner of the substrate's bottom face (that is, the +X, -Y, -Z corner of the model) .
    b. Click the top right corner of the substrate's bottom face (that is, the -X, +Y, -Z corner of the model).
    c. Click an arbitrary third point above the model to define the air body's height.
2. In the **Command** tab of the *Properties* dialog box that appears, specify a **ZSize** value of **600 µm**, as shown in the following figure:

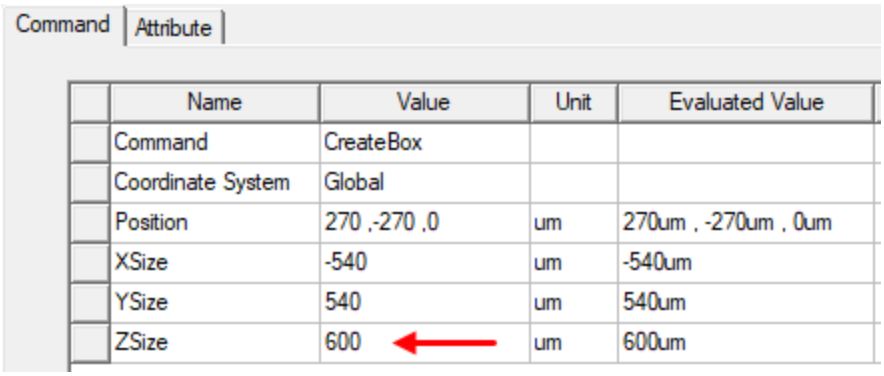



**Figure 3-13: Air Body Properties – *Command* Tab**

3. On the **Attribute** tab, make the following changes:
    a. Change the object **Name** to **Air**.
    b. Select **Edit** from the **Material** drop-down menu, choose **air** from the listed materials in the *Select Definition* dialog box, and click **OK**.
    c. Ensure that the **Material Appearance** option is **not** selected.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Set the **Color** to **medium gray** (column 4, row 6 of the *Basic color* samples; Red: 128, Green: 128, Blue: 128).
e. Set the **Transparent** value at **0.8**.

4. Click **OK** to close the Properties dialog box.
5. Press **Ctrl+D** to fit the view. Also, clear the selection.

   The completed dielectric objects should look like the following image:

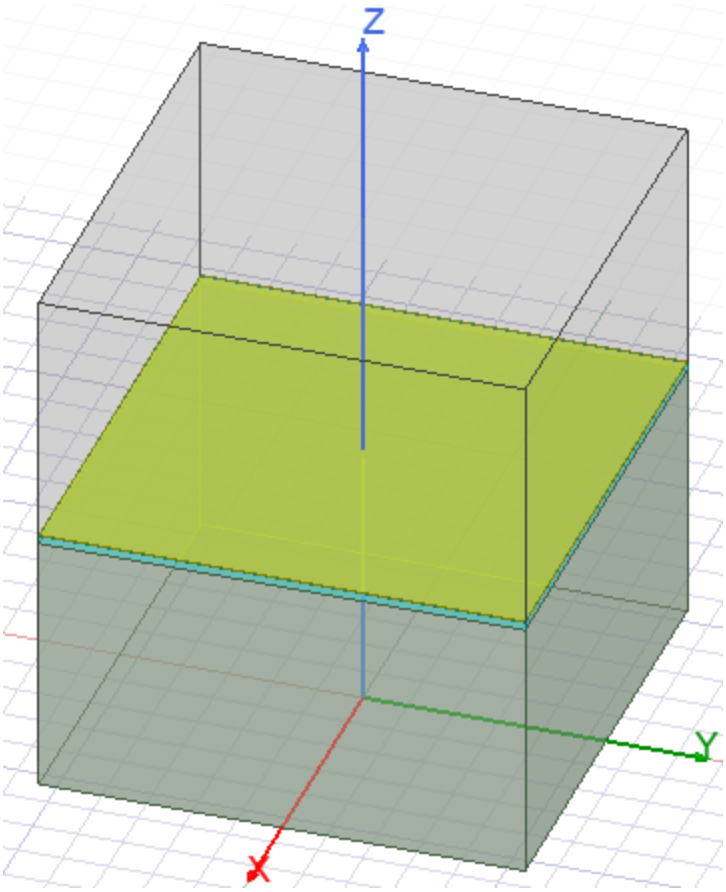



**Figure 3-14: Air Body Created**

6. **Save** your project. (This command is available from any ribbon tab.)

# Create Conductors

You will begin the conductors by creating a ground plane at the base of the substrate. Next, hide the dielectric objects and ground plane to facilitate creation of the remaining conductors. You will then draw the remaining conducting objects (spiral inductor assembly and ground ring). The following topics and subsections are covered in this section:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Create Ground Plane
- Hide Dielectric Objects
- Create Spiral Inductor
- Create Ground Ring

# Create Ground Plane

You will create the ground plane object from the bottom face of the substrate, as described below:

1. Ensure that the Modeler window is the active window by clicking anywhere inside it. Then, press **F** to switch to the *face selection* mode.
2. Click near the bottom edge of one of the substrate's side faces. Then, press **B** (*Next Behind*) twice to select the bottom face.
3. On the **Draw** ribbon tab, click **Surface** > **Create Object From Face**.

   *Air_ObjectFromFace1* appears under *Model* > *Sheets* > *Unassigned* in the History Tree, and its attributes appear in the docked Properties window.

4. In the **Attribute** tab of the docked Properties window, make the following changes:
   a. Change the **Name** to **Ground** and press **Enter**.
   b. Change the **Color** to **orange** (column 2, row 4 of the *Basic color* samples; Red: 255, Green: 128, Blue: 0).
   c. Set the **Transparent** value to **0** (opaque).
5. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

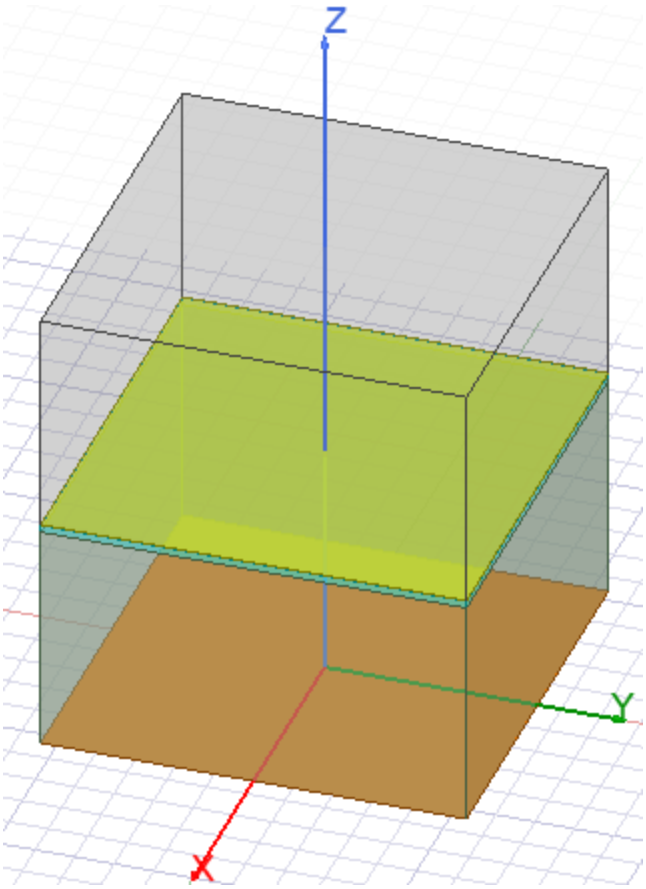



**Figure 3-15: Ground Plane Added**

# Hide All Existing Objects

Next, hide the existing objects to facilitate drawing the remaining conductors.

1. Ensure that the Modeler window is the active window by clicking anywhere inside it. Then, press **O** to return to the *object selection* mode.
2. Press **Ctrl+A** to select all objects in the window.

   Alternatively, you can use the menu bar to click **Edit** > **Select All** or **Edit** > **Select All Visible**. You can also right-click in the Modeler window and choose **Select Objects** > **All Model Objects** from the shortcut menu.

3. On the **Draw** ribbon tab, click **Hide selected objects in active view**.

   Alternatively, you can use the menu bar to click **View** > **Visibility** > **Hide Selection** > **All Views** (or **Active Views**). You can also right-click in the Modeler window and choose


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

**View** > **Hide Selection** from the shortcut menu.

All objects created thus far are now hidden.

# Create Spiral Inductor

The spiral inductor starts out as a polyline following a spiral path. Two methods of drawing the path are provided. You next define the width and thickness of the conductor and create the underpass, vias, and feed object. Finally, these objects are united into the complete spiral inductor object.

The following topics are included in this subsection:

- Define Conductor Material
- Create Offset Coordinate System
- Create Spiral Path
- Assign Width and Thickness
- Create Underpass
- Create Via 1 and Via 2
- Create Feed
- Unite Spiral Objects

## Define Conductor Material

Before you create the conductors, define a new default material.

1. At the far right end of the Draw ribbon tab, choose **Select** from the **Default material** drop-down menu. (There is no icon associated with this command, and the current default material is shown, most likely *vacuum*.)

   The *Select Definition* dialog box appears.

2. Click **Add Material**.

   The *View/Edit Material* dialog box appears.

3. Edit the **Material Name** and **Bulk Conductivity** value as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

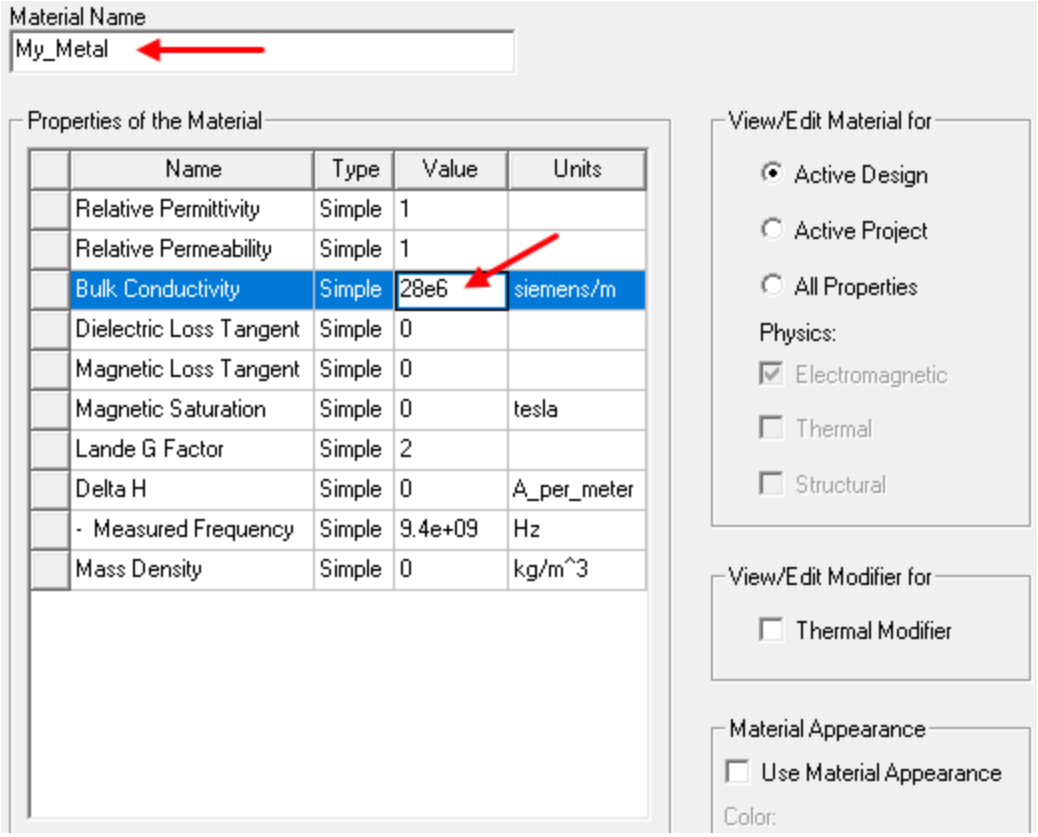



**Figure 3-16: View/Edit Material dialog box**

4. Click **OK** twice to close the *View / Edit Material* and *Select Definition* dialog boxes.

   *My_Metal* is now shown on the *Draw* ribbon tab as the current *Default material*.

## Define an Offset Coordinate System

Create an offset coordinate system (with the drawing plane parallel to the global XY plane) with a Z offset at the desired spiral path elevation. The spiral path is located within the thickness of the Oxide layer at Z = 305.8 µm.

1. On the **Draw** ribbon tab, click **Relative CS**. (You do not have to access the *Relative CS* drop-down menu to create an offset CS, since *Offset* is the default action when you click the *Relative CS* icon.)

   The message *Select the origin* appears at the left end of the status bar.

2. Press **Tab** to jump to the **X** coordinate text box (the first of three near the right end of the status bar). Specify the origin coordinates for the Offset CS as follows:
   - Type **0** in the **X** text box and press **Tab**.
   - Type **0** in the **Y** text box and press **Tab**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Type **305.8** in the **Z** text box and press **Enter**.

> **Note:**
>
> Be careful not to move your mouse while entering coordinates in the text boxes, or the cursor location will override the coordinates you specify.

*RelativeCS1* appears under *Coordinate Systems* in the History Tree, and it is the working (active) CS, as indicated by the "w" on the icon:

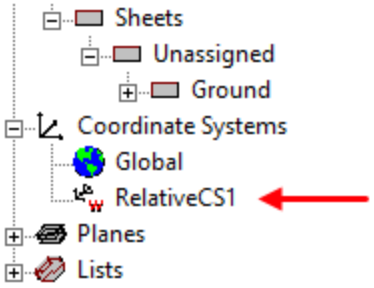



**Figure 3-17: *RelativeCS1* Defined**

3. **Save** your project.

## Create Spiral Path

In this procedure, you will create a spiral polyline path consisting of twelve (12) straight line segments. You will specify the endpoint coordinates numerically in the status bar's coordinate text boxes while drawing the polyline.

> **Note:**
>
> Since *RelativeCS1* is already defined and is the working CS, you will use it for drawing the polyline. Since the Z elevation is set via the relative coordinate system's Z offset, the specified Z coordinate will be zero for all points (rather than 304.8 um).
>
> It is possible to use *Global* as the working coordinate system and then associate the geometry with *RelativeCS1* later on, which would correct the Z elevation of the spiral. However, that approach would be less efficient.

1. On the **Draw** ribbon tab, click **Draw line**.
2. Use the **Tab** key to navigate among the **X, Y,** and **Z** coordinate entry text boxes in the status bar. Specify the **X** and **Y** coordinates from the following table for each point along the polyline path. The **Z** coordinate is zero for all points. The global Z elevation is

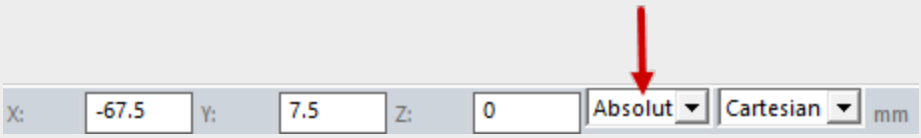

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

predefined according to the offset coordinate system (*RelativeCS1*). Press **Enter** when each set of coordinates is complete to draw the segment and advance to the next point.

Proceed cautiously, as there is considerable room for entry errors. You may want to print a hard copy of this topic so that you can check off the points as you enter them.

**Important:**

- Ensure that the drop-down menu to the right of the coordinate text boxes is set to **Absolute** for the first point and for all subsequent points:




- Keep your hand off of the mouse when tabbing into the coordinate text boxes and be very careful *not* to bump the mouse while typing the coordinates. Any mouse movement will cause the numerical values to revert to the graphical location of the cursor, resulting in an incorrect line segment.

| Point Number | X: | Y: | Z: |
|---|---|---|---|
| **1** | **-67.5** | **7.5** | 0 |
| **2** | **-67.5** | **-67.5** | 0 |
| **3** | **84** | **-67.5** | 0 |
| **4** | **84** | **84** | 0 |
| **5** | **-84** | **84** | 0 |
| **6** | **-84** | **-84** | 0 |
| **7** | **100.5** | **-84** | 0 |
| **8** | **100.5** | **100.5** | 0 |
| **9** | **-100.5** | **100.5** | 0 |
| **10** | **-100.5** | **-100.5** | 0 |
| **11** | **117** | **-100.5** | 0 |
| **12** | **117** | **0** | 0 |
| **13** | **131** | **0** | 0 |


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Right-click in the Modeler window and select **Done** from the short-cut menu to terminate the polyline at the thirteenth point.

   The *Properties* dialog box appears.

4. In the **Attribute** tab of the Properties dialog box, make the following changes:
   a. Change the **Name** to **Spiral** and press **Enter**.
   b. Set the **Color** to **red** (column 1, row 2 of the *Basic colors* samples; Red: 255, Green: 0, Blue: 0).
   c. Ensure that the **Transparent** value is **0** (opaque).
5. Click **OK** to close the *Properties* dialog box.
6. Press **Ctrl+D** to fit the view. Also, clear the selection.

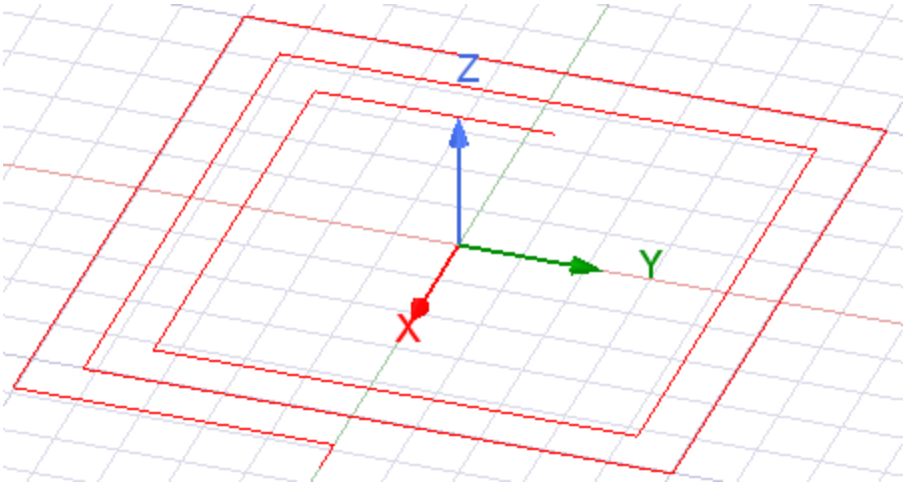



**Figure 3-18: Spiral Path Drawn**

The preceding image was captured using the **Small** coordinate system view option so that the X axis would not obscure the final, short polyline segment.

7. If your spiral path does not look correct, select each of the **CreateLine** entries under *Model* > *Lines* > *Spiral* > *CreatePolyline* in the History Tree and verify the coordinates against the preceding table. Correct any values that were specified incorrectly.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Each polyline segment (that is, each *CreateLine* entry in the History Tree) has two endpoints (Point 1 and Point 2) defined in the *Segment* tab of the docked Properties window. Therefore, the point numbering will not match the preceding table. For each segment, *Point 1* is the same as *Point 2* of the preceding segment. Likewise, *Point 2* is the same as *Point 1* of the next segment. Changing the coordinates for one segment updates the corresponding point in the adjacent segment.

8. **Save** your project.

## Assign Width and Thickness to Spiral

To assign a width and thickness to the spiral polyline path, perform the following steps:

1. Under *Model* > *Lines* > *Spiral* in the History Tree, select **CreatePolyline**.

   The polyline settings appear in the *Command* tab of the docked *Properties* window.

2. In the *Cross Section* part of the *CreatePolyline* properties, click in the *Value* column of the **Type** row and select **Rectangle** from the drop-down menu.

   Additional items appear in the docked Properties window.

3. Specify the following settings:
   a. **Width/Diameter** = **15 µm**
   b. **Height** = **2 µm**
   c. Press **Enter**.

| Name | Value | Unit | Evaluated Value |
|---|---|---|---|
| Command | CreatePolyline | | |
| Coordinate System | RelativeCS1 | | |
| Number of points | 13 | | |
| Number of curves | 12 | | |
| **Cross Section** | | | |
| Type | Rectangle | | |
| Orientation | Auto | | |
| Width/Diameter | 15 | um | 15um |
| Height | 2 | um | 2um |
| Bend Type | Corner | | |


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

**Figure 3-19: Polyline Cross Section Properties**

The spiral is assigned the width and thickness that you specified.

4. Press **Ctrl+D** to fit the view. Also, clear the selection.

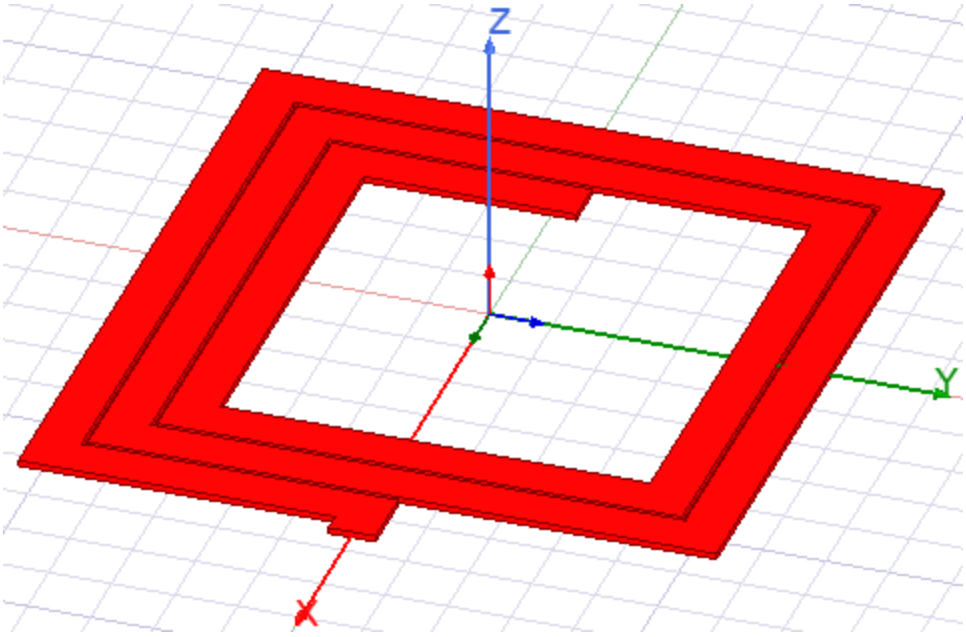



**Figure 3-20: Updated Spiral**

## Create Underpass

The underpass is a conductor that is 75 µm long x 15 µm wide x 0.5 µm thick. There needs to be a gap of 0.8 µm between the bottom face of the spiral and the top face of the underpass. Since the spiral is 2 µm thick, its bottom face is 1 µm below the *RelativeCS1* drawing plane. Therefore, the top of the underpass must have a Z elevation of -1.8 µm. You will draw a box with this Z coordinate in its *Position* value and with a *ZSize* of -0.5 µm.

1. Ensure that *RelativeCS1* is still the working coordinate system:

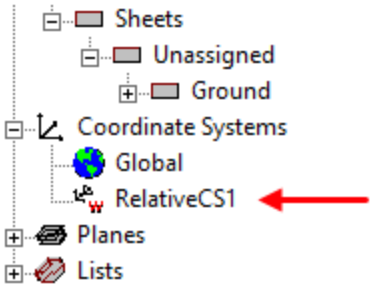



**Figure 3-21: Verifying Working Coordinate System (*W* on Icon)**

2. On the **Draw** ribbon tab, click **Draw box**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click three different points to draw an box with an arbitrary size and location.

   After third click, the *Properties* dialog box appears.

4. In the **Command** tab of the *Properties* dialog box, specify the settings shown in the following figure:

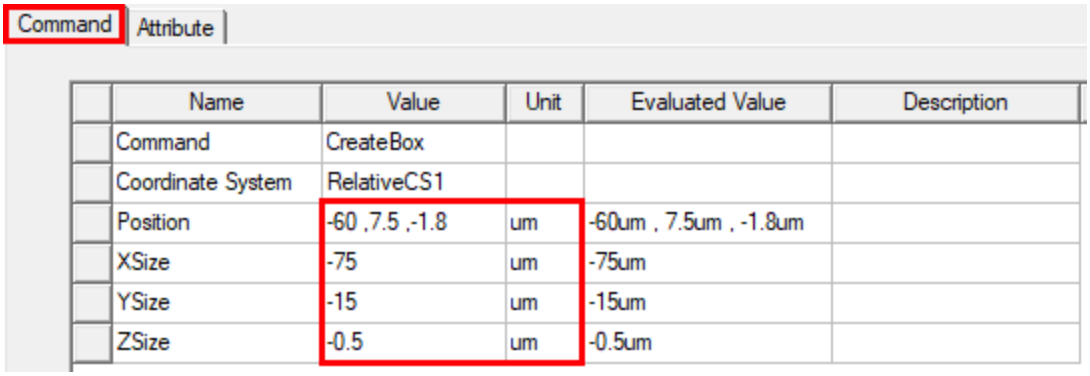



**Figure 3-22: Underpass Properties – *Command* Tab**

5. On the **Attribute** tab, change the object **Name** to **Underpass** and click **OK** to close the *Properties* dialog box.

> **Note:**
>
> The object appearance settings (color and transparency) do not matter. In a later step, you will unite the objects comprising the spiral inductor assembly. At that time, all objects will assume the material and appearance of the first object selected for the *Unite* operation.

6. Clear the selection.

   Your model should look like the following image:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

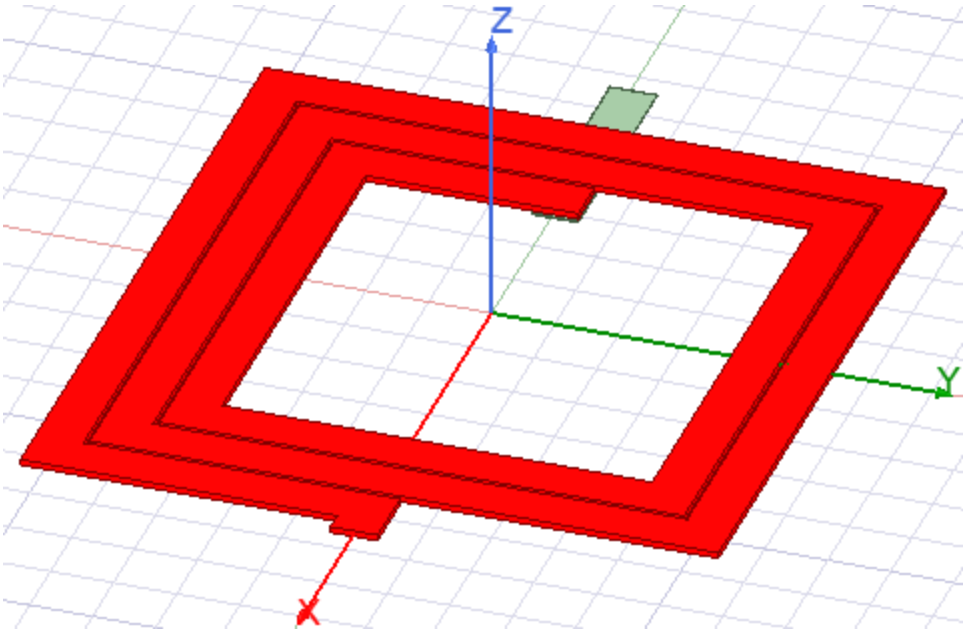



**Figure 3-23: Underpass Created**

# Create Vias

You will create two vias, one at each end of the underpass. *Via 1* connects the spiral inductor to the underpass. *Via 2* connects the underpass to the feed (not yet drawn).

If the first two points you specify when drawing a box differ in all three coordinates (X, Y, and Z), the length, width, and height are defined with only those two points, and a third click is not needed. You will take advantage of that functionality, and two snap points on existing geometry, to draw the first via. In order to click a second point that's not on the same plane as the first one, you must switch from the *In Plane* to the *3D* movement mode while drawing the box.

**Create *Via 1*:**

Draw a box as follows:

1. Zoom in closely to the area where the spiral will connect to the underpass. The gap between these objects is small, and a tight zoom area is helpful in snapping to the correct vertices.
2. On the **Draw** ribbon tab, click **Draw box**.
3. Click the vertex indicated in the following image:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

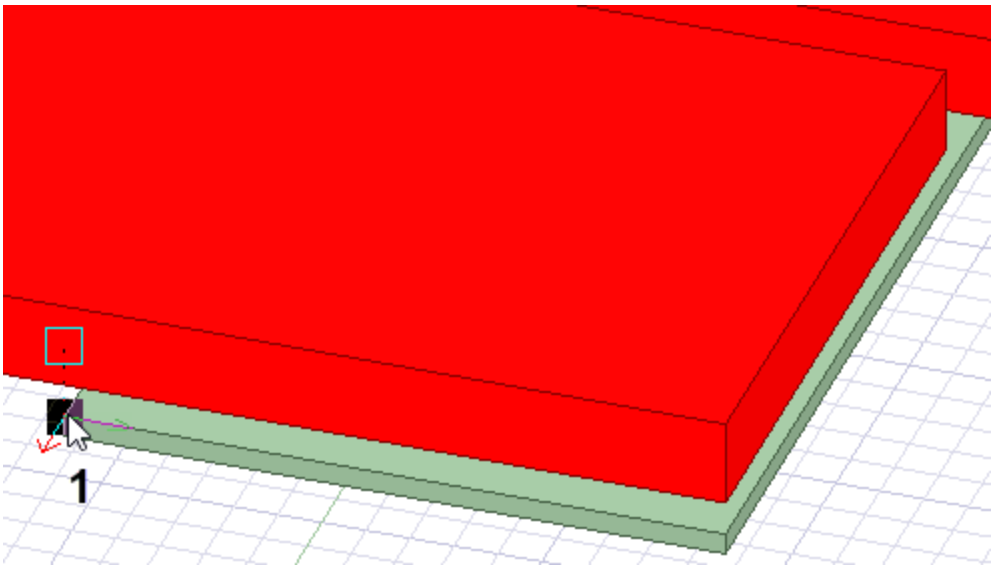



**Figure 3-24: First Point of *Via 1* Box**

4. Right-click in the Modeler window and select **Movement Mode** > **3D** from the shortcut menu.
5. Click the point indicated in the following figure to simultaneously define the length, width, and height of the box:

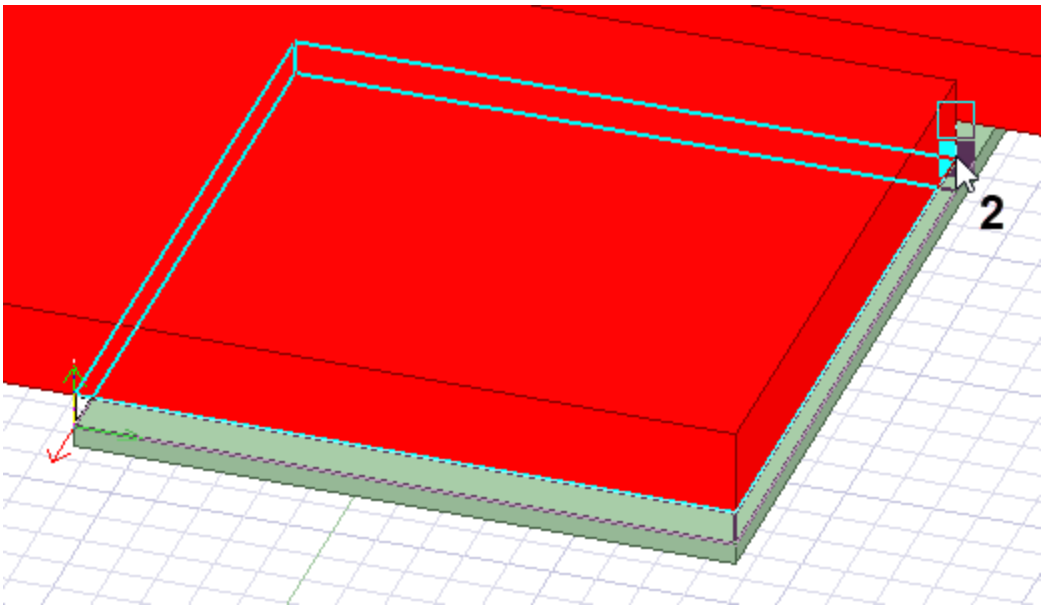



**Figure 3-25: Second Point of *Via 1* Box (3D Drawing Mode)**

After the second click, the *Properties* dialog box appears.

6. On the **Attribute** tab of the Properties dialog box, change the **Name** to **Via1** and then click **OK**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Clear the selection.

   The model should now look like the following figure:

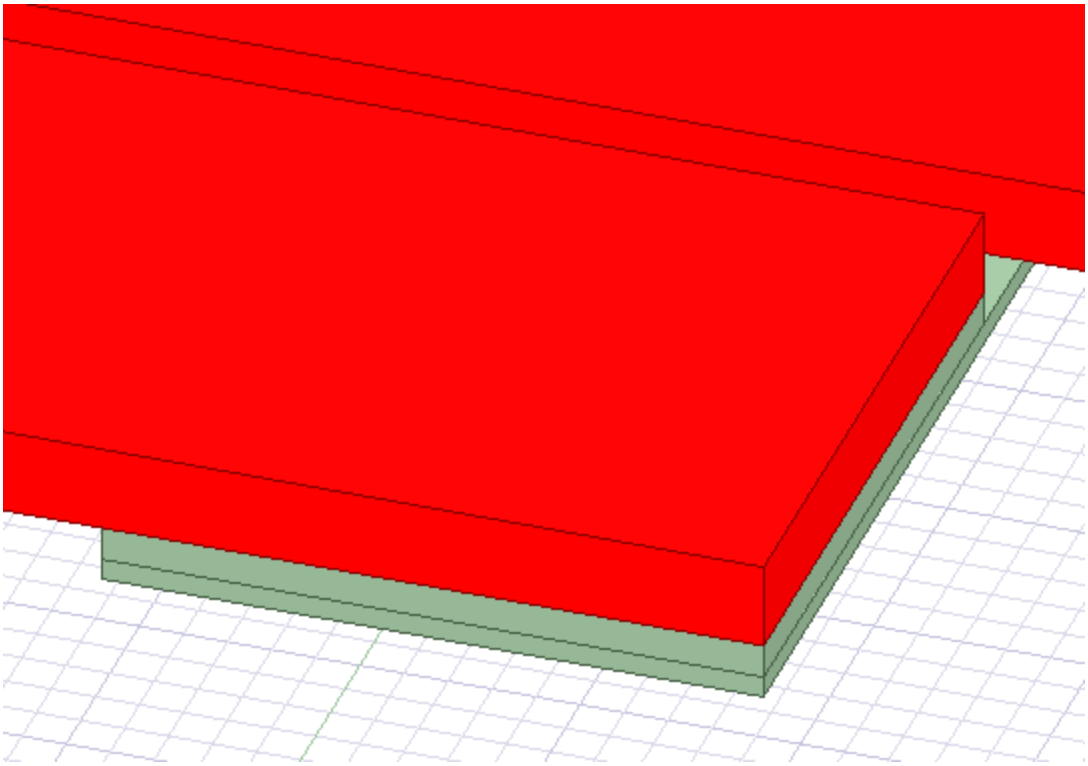


**Figure 3-26: Via 1 Created**

**Create Via 2:**

Draw another box using one existing snapping point and enter a second point in the coordinate entry text boxes, as follows:

8. Pan the model view to see the opposite end of the underpass.
9. On the **Draw** ribbon tab, click **Draw box**.
10. Click the point indicated on the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

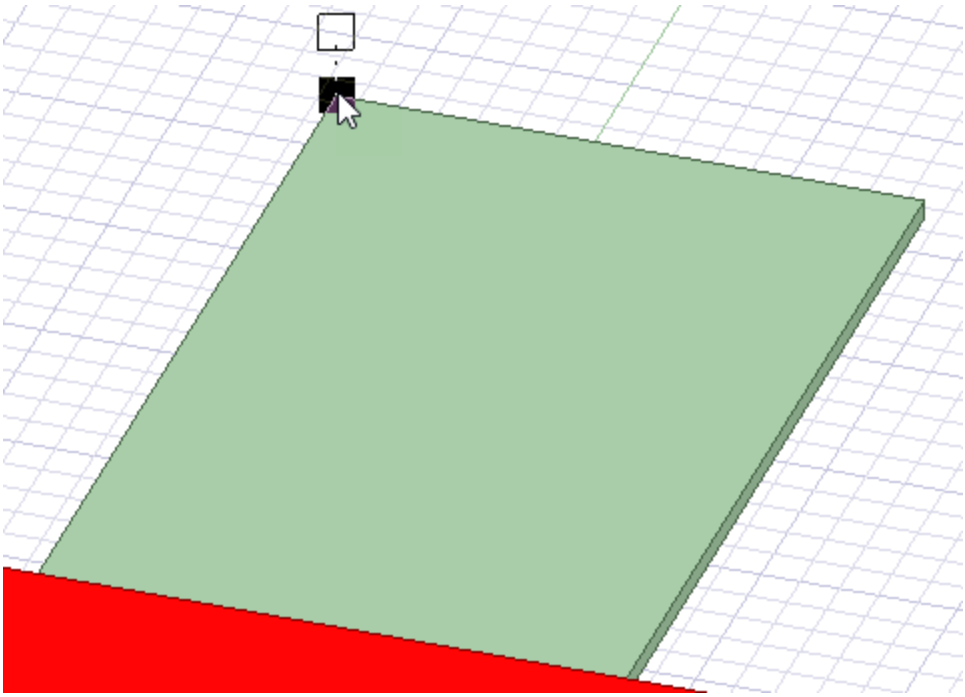


**Figure 3-27: First Point of Via 2 Box**

11. Type the following coordinates in the status bar's coordinate entry text boxes and then press **Enter**:
    - **dX: 15**
    - **dY: 15**
    - **dZ: 0.8**
12. On the **Attribute** tab of the Properties dialog box, change the **Name** to **Via2** and click **OK**.
13. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

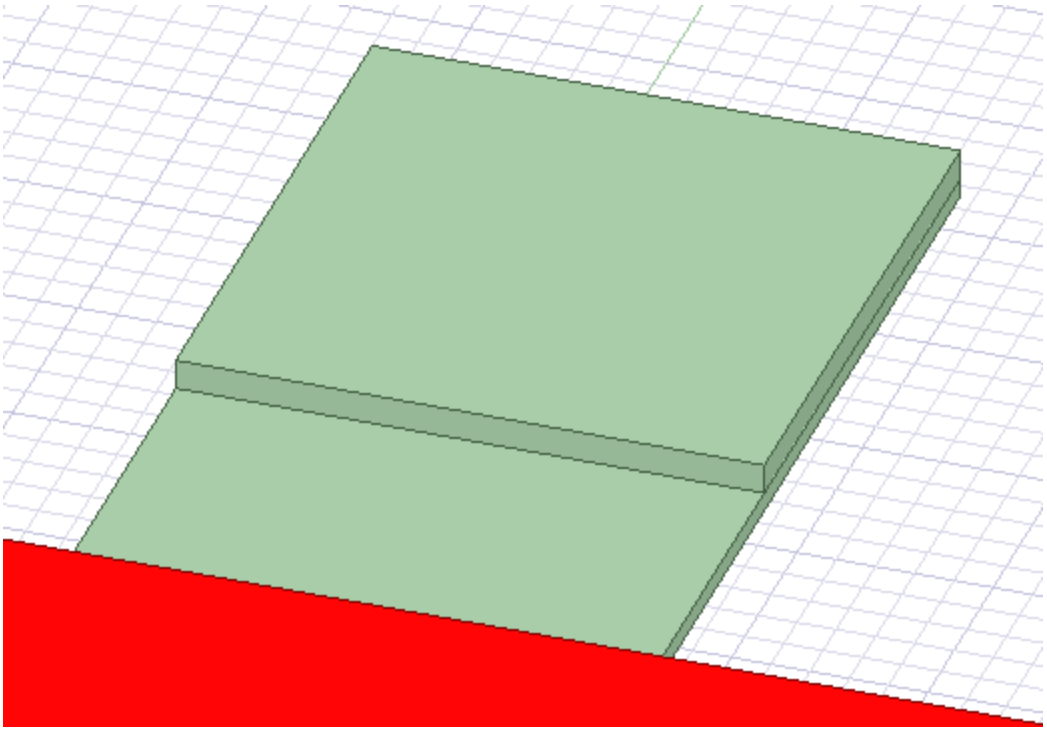


**Figure 3-28: Via2 Created**

## Create Feed

Draw the Feed object in the same way that you drew Via 2:

1. On the **Draw** ribbon tab, click **Draw box**.
2. Click the point indicated on the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

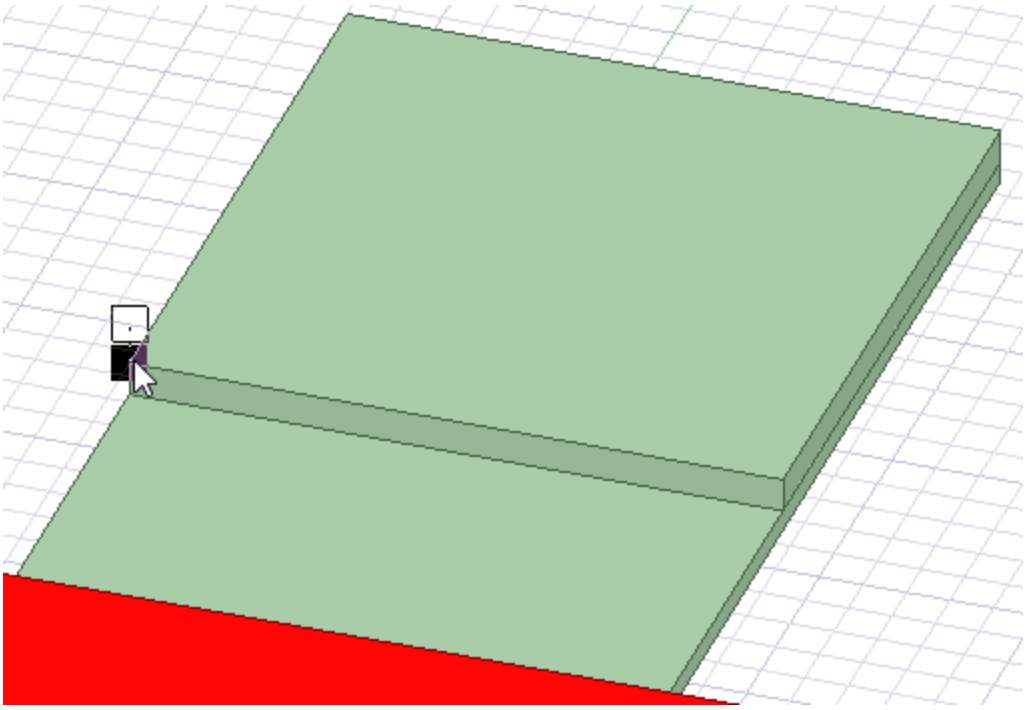


**Figure 3-29: First Point of Feed Box**

3. Type the following coordinates in the status bar's coordinate entry text boxes and then press **Enter**:
   - **dX: -22**
   - **dY: 15**
   - **dZ: 2**
4. On the **Attribute** tab of the Properties dialog box, change the **Name** to **Feed** and click **OK**.
5. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

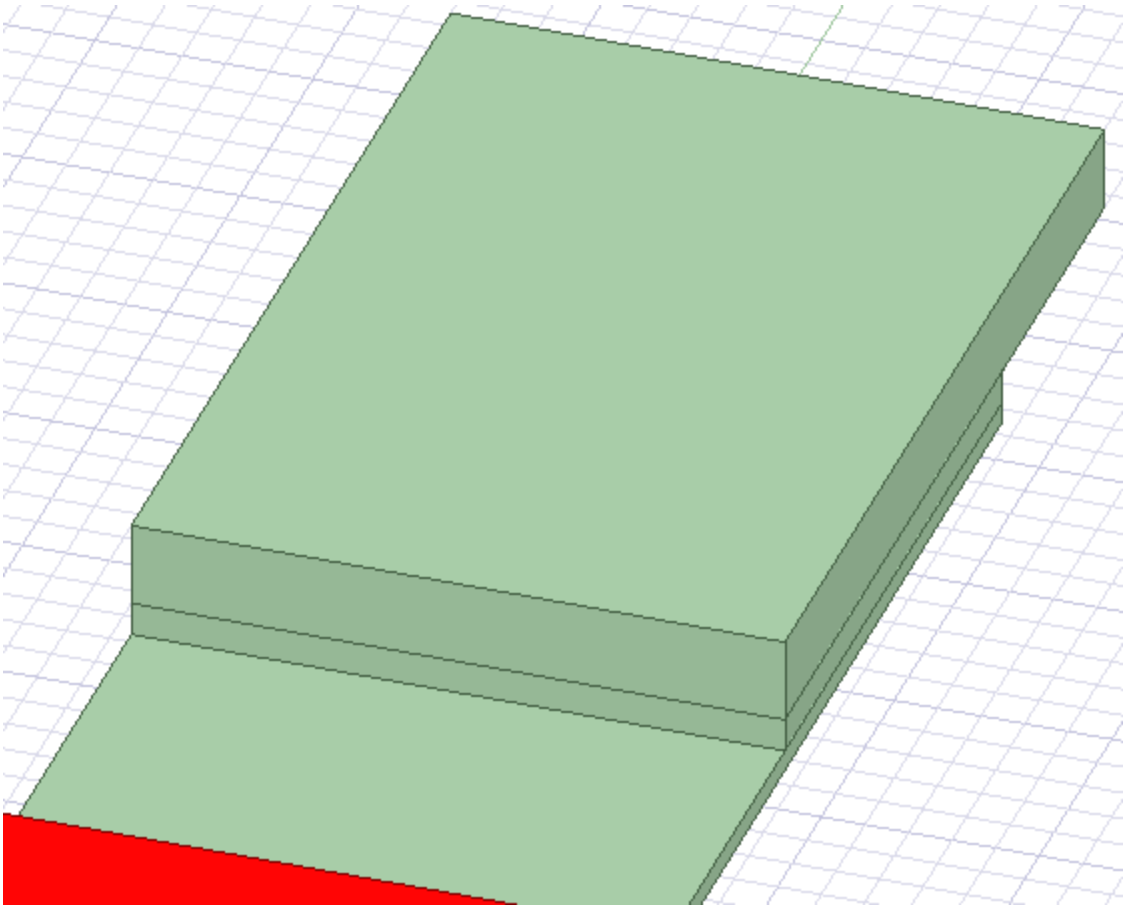


**Figure 3-30: Feed Created**

## Unite Spiral Objects

You will now unite the spiral inductor objects.

1. Press **Ctrl+D** to fit the visible objects to the viewing area.
2. Under *Model* > *Solids* > *My_Metal* in the History Tree, select **Spiral**.
3. Hold down the **Ctrl** key and also select **Feed, Underpass, Via1,** and **Via2**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

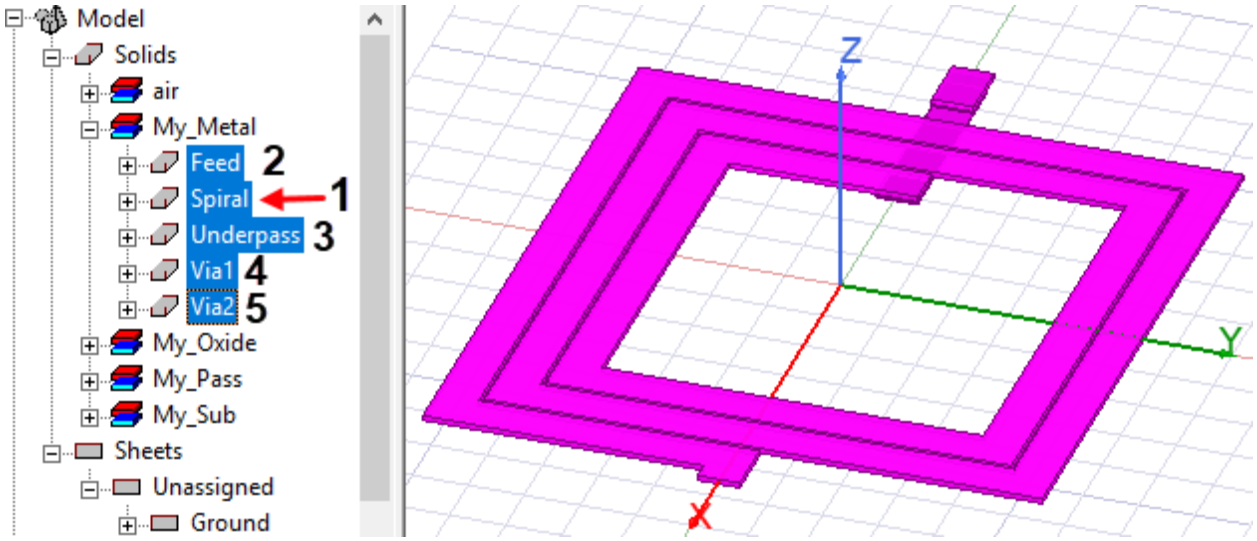



**Figure 3-31: Selecting Objects to Unite**

> **Note:**
>
> The order in which you select the objects determines the name of the united structure, the material, and the material appearance. By selecting the **Spiral** object first, the united structure will be named *Spiral* and will have all of the *Spiral* attributes. If you were to select **Feed** first, the united structure would be named *Feed* and have that objects attributes.

4. On the **Draw** ribbon tab, click **Unite**.
5. Clear the selection.

   There is now only one object, *Spiral*, under *My_Metal* in the History Tree, and the model appearance should be as shown below:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

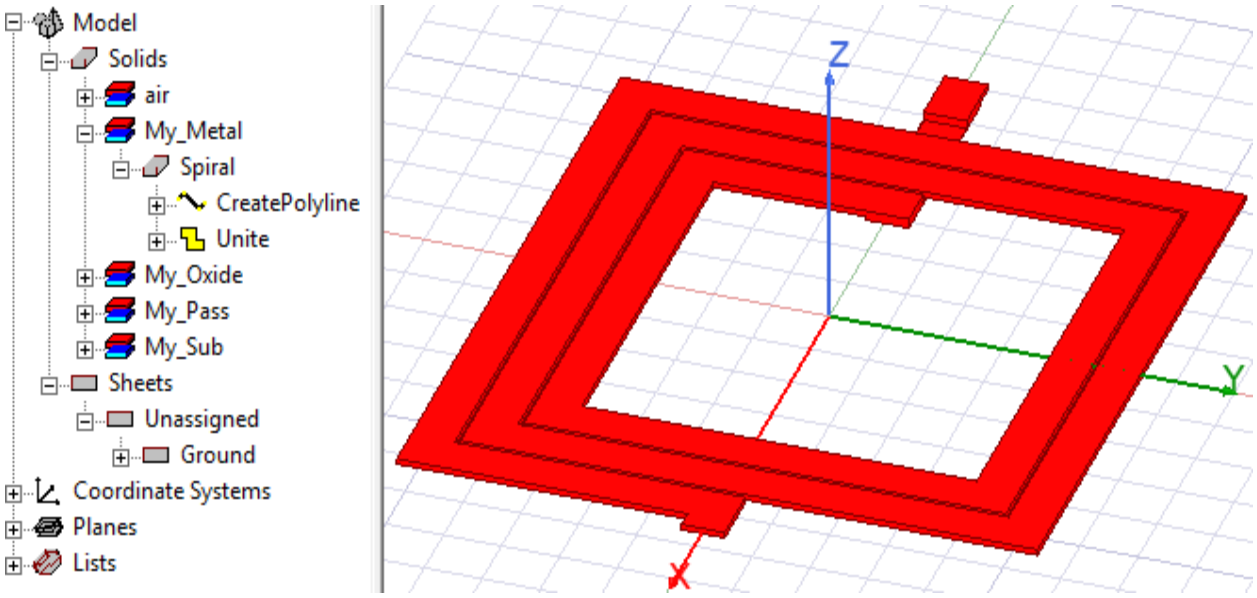



**Figure 3-32: United Spiral Object**

> **Note:**
>
> The conductive material is represented by a boundary condition that eliminates the need to solve inside the metal.

6. **Save** your project.

# Create Ground Ring

A ground ring surrounds the spiral conductor. Before drawing the ground ring, you will create a second offset coordinate system and define a new default material. You will then draw two boxes, a large box and a smaller one within it. You will create the ring by subtracting the smaller box from the larger one. Finally, you will add extensions for connecting the ground ring to the spiral inductor sources and then unite the objects.

This subsection consists of the following topics.

- Create a Second Offset CS
- Define Ground Ring Material
- Create Outer Box
- Create Inner Box
- Complete the Ring


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Create Extensions
- Unite Ring and Extensions

## Create a Second Offset CS

To facilitate drawing of the ground ring and its extensions, you will define a second offset coordinate system.

*RelativeCS1* has a Z coordinate (elevation) that corresponds to the middle of the thickness of the spiral inductor and feed. You drew the spiral polyline, which was the basis of the inductor, at the middle of its planned thickness. Then, you applied the width and thickness to the polyline after drawing it. You will draw the ground ring and its extensions as boxes. For the elevation of the base rectangles to correspond with the elevation at the bottom face of the spiral and feed, the new coordinate system must have a *-1 µm* Z-coordinate relative to *RelativeCS1*.

1. Ensure the *RelativeCS1* is still the working coordinate system.
2. On the **Draw** ribbon tab, click **Relative CS**. (You do not have to access the *Relative CS* drop-down menu. *Offset* is the default action when you click the *Relative CS* icon.)

   The message *Select the origin* appears at the left end of the status bar.
3. Press **Tab** to jump to the **X** coordinate text box. Specify the origin coordinates for the new Offset CS as follows:
   - Type **0** in the **X** text box and press **Tab**.
   - Type **0** in the **Y** text box and press **Tab**.
   - Type **-1** in the **Z** text box and press **Enter**.

   > **Note:**
   >
   > Be careful not to move your mouse while entering coordinates in the text boxes, or the cursor location will override the coordinates you specify. The absolute coordinates you enter are based on the working coordinate system, not the *Global* one.

   *RelativeCS2* appears under *Coordinate Systems* in the History Tree, and it is the working CS, as indicated by the "w" on the icon:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

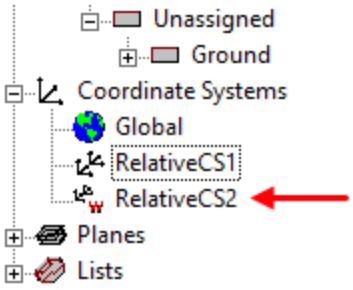



**Figure 3-33: *RelativeCS2* Defined**

# Define Ground Ring Material

1. On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu. (*My_Metal* is currently shown as the default material.)

   The *Select Definition* dialog box appears.

2. Type **pec** in the **Search by Name** text box.

   In the list of library materials, *pec* (perfect electrical conductor) is selected.

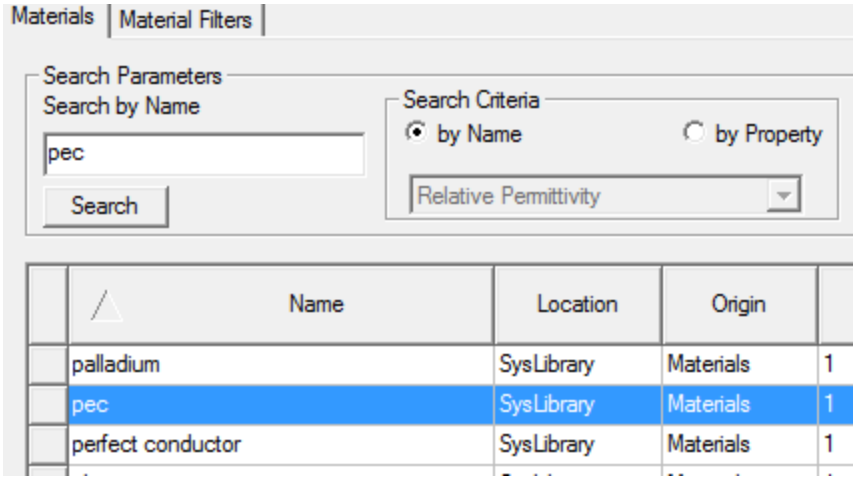



**Figure 3-34: Specifying Ground Ring Material**

3. Click **OK** to close the dialog box.

   All new objects you draw will be assigned the pec material until the default is changed again.

# Create Outer Box

To create a ground ring, first draw the outer box freehand.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **Draw** ribbon tab, click **Draw box**.
2. Click at three random points to draw a box of arbitrary size and location.

   The *Properties* dialog box appears.

3. On the **Command** tab of the *Properties* dialog box, edit the values as shown in the following figure:

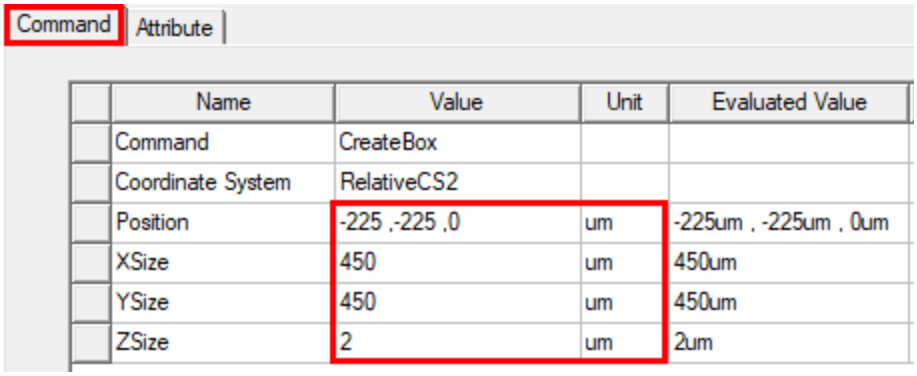



**Figure 3-35: Outer Box Properties – *Command* Tab**

4. On the **Attribute** tab, make the following changes:
   a. Change the **Name** to **GND_Ring** and press Enter.
   b. Ensure that the **Material Appearance** option is **not** selected.
   c. Set the **Color** to **orange** (column 2, row 4 of the *Basic color* samples; Red: 255, Green: 128, Blue: 0).
   d. Set the **Transparent** value at **0** (opaque).
5. Click **OK** to close the *Properties* dialog box.
6. Press **Ctrl+D** to fit the view.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

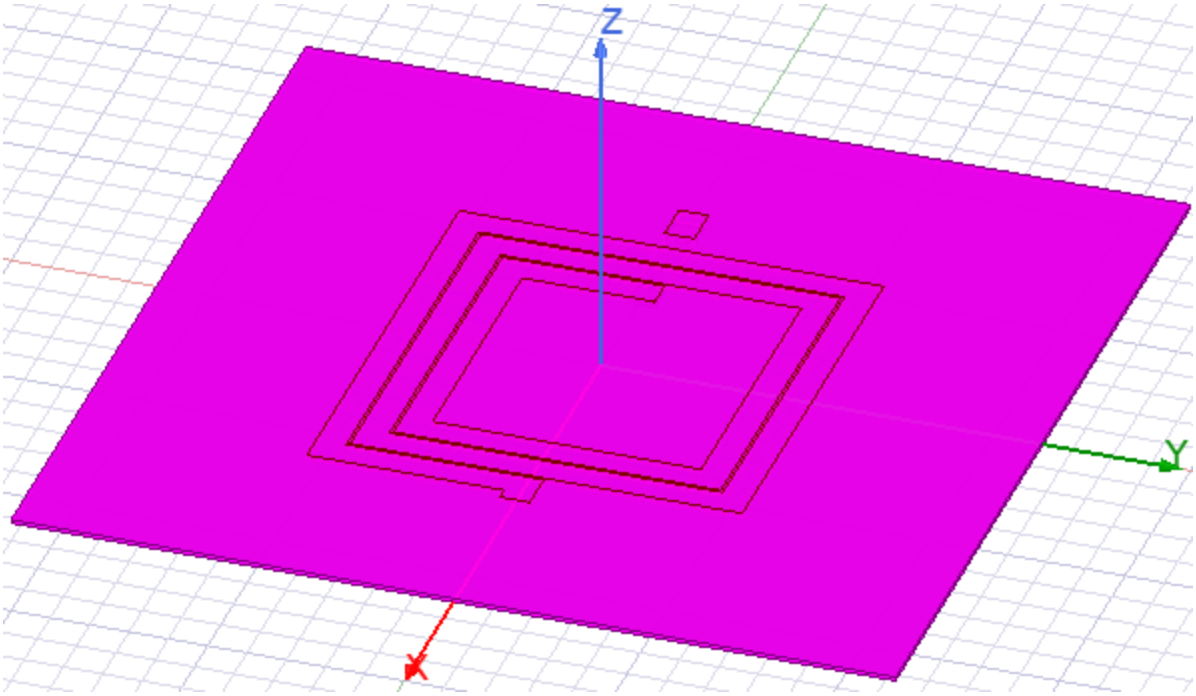



**Figure 3-36: Outer Box Drawn**

## Create Inner Box

You will create an inner box to be subtracted from the outer box you just drew. The result will be a rectangular ring surrounding the spiral inductor.

To create the inner box, do as follows:

1. On the **Draw** ribbon tab, click **Draw box**.
2. Click at three random points to draw a box of arbitrary size and location.

   The *Properties* dialog box appears.

3. On the **Command** tab of the *Properties* dialog box, edit the values as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command | Attribute

| | Name | Value | Unit | Evaluated Value |
|---|---|---|---|---|
| | Command | CreateBox | | |
| | Coordinate System | RelativeCS2 | | |
| | Position | -210 ,-210 ,0 | um | -210um , -210um , 0um |
| | XSize | 420 | um | 420um |
| | YSize | 420 | um | 420um |
| | ZSize | 2 | um | 2um |

**Figure 3-37: Inner Box Properties – *Command* Tab**

4. On the **Attribute** tab, change the **Name** to **InnerBox** and click **OK**.

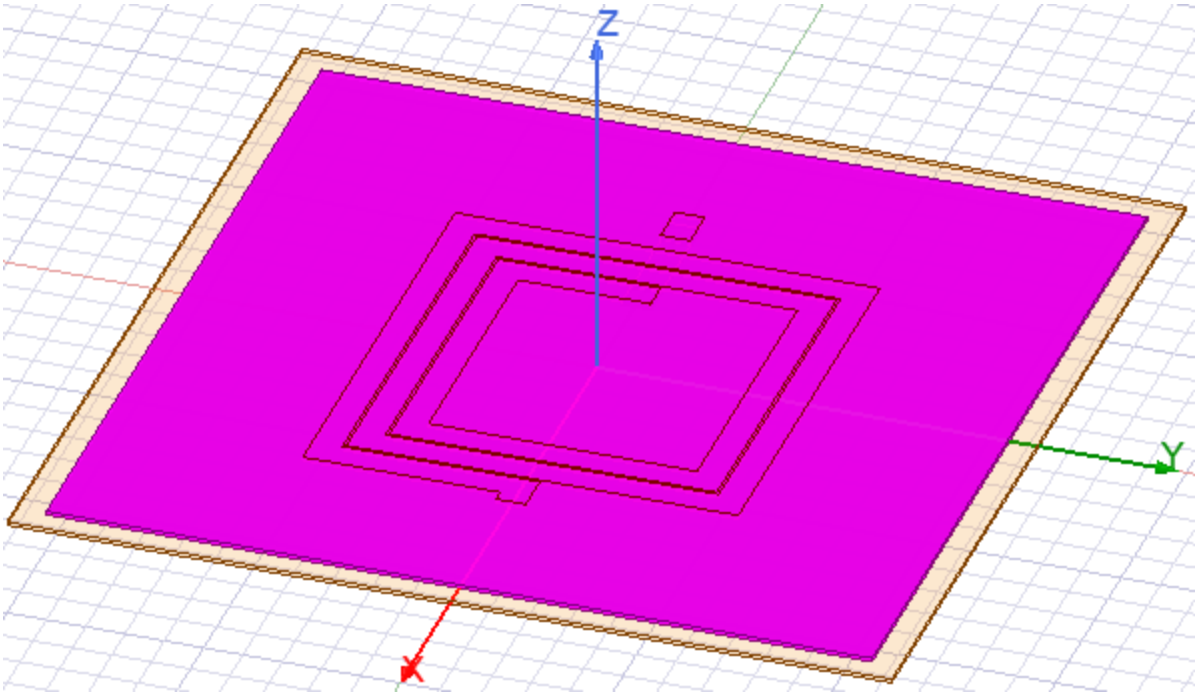



**Figure 3-38: Inner Box Drawn**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The appearance attributes of this object do not matter. It will only be used as a cutting tool to produce a ring from the outer box. After subtraction, this object will no longer be present in the model (except as an entry in the History Tree under the *Subtract* operation).

## Complete the Ring

You will now subtract the inner box from the outer box to produce the desired ring shape.

1. While the **Object** *selection mode* is active, on the **Draw** ribbon tab, click **Select by Name**.

   The *Select Object* dialog box appears.

2. Select **GND_Ring** and hold **Ctrl** while clicking **InnerBox** to select it too. Then, click **OK**.

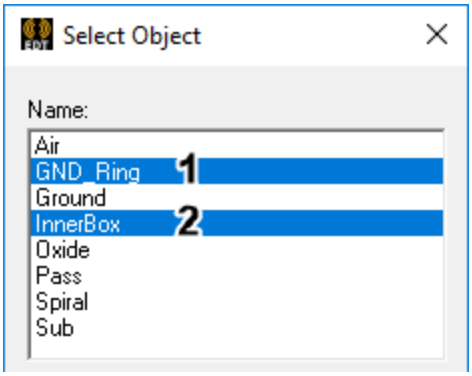



**Figure 3-39: Selecting Objects for Subtraction**

3. On the **Draw** ribbon tab, click **Subtract**.

   The *Subtract* dialog box appears.

4. Verify that **GND_Ring** is in the *Blank Parts* box, and **Inner** in the *Tool Parts* box. Then, click **OK**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

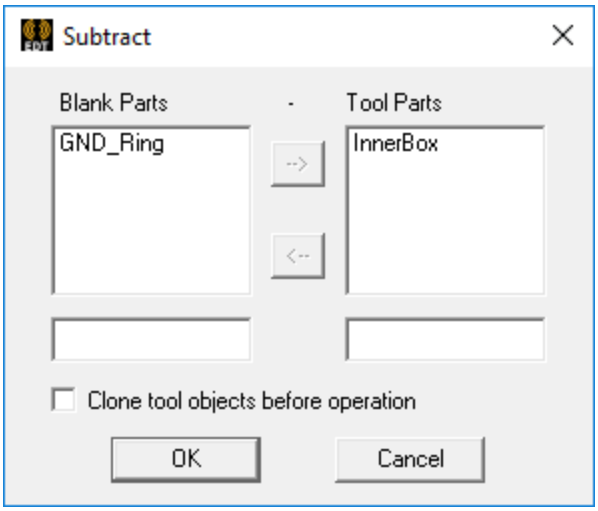



**Figure 3-40: *Subtract* Dialog Box**

5. Clear the selection.

   The model should resemble the following figure:

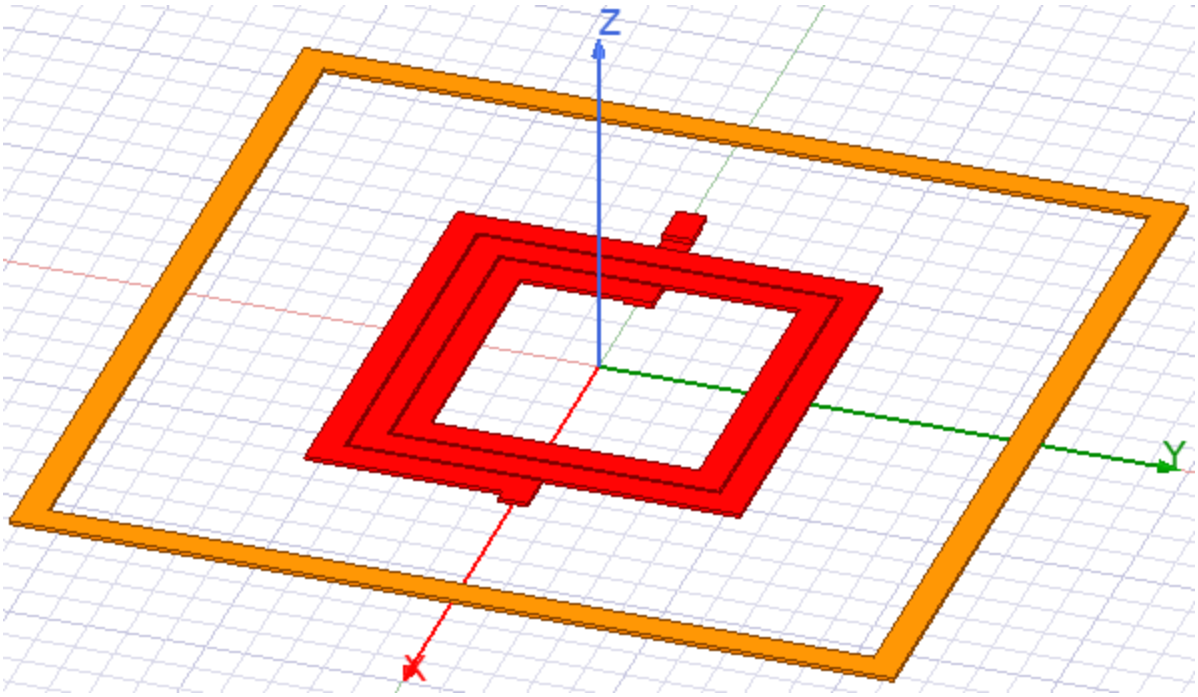



**Figure 3-41: Ground Ring Created**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Create Extensions

You will create two extensions on the inside of the ring for connecting to the sources at each end of the spiral inductor.

**Draw the first extension as follows:**

1. Draw a box freehand.

   The *Properties* dialog box appears after you click the third point.

2. On the **Command** tab, edit the values as shown below:

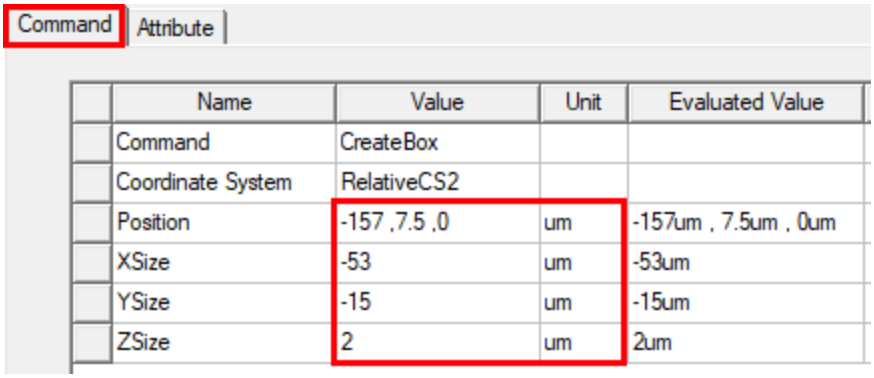



**Figure 3-42: Extension 1 Properties – *Command* Tab**

3. On the **Attribute** tab change the object **Name** to **Ext1** and click **OK**.

> **Note:**
>
> The material appearance attributes do not matter. Once united with the ground ring, the extensions will assume the same attributes as the ring.

4. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

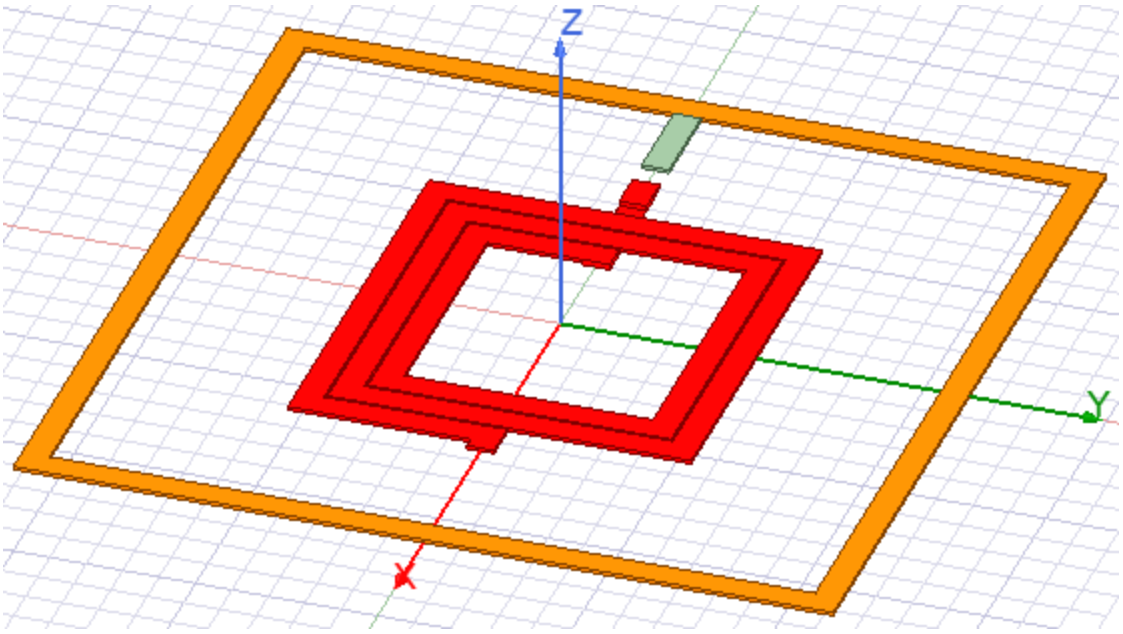



**Figure 3-43: Extension 1 Added to Ground Ring**

**Draw the second extension as follows:**

5. Draw a box freehand.

   The *Properties* dialog box appears.

6. In the **Command** tab, edit the values as shown below:

Command | Attribute

| | Name | Value | Unit | Evaluated Value |
|---|---|---|---|---|
| | Command | CreateBox | | |
| | Coordinate System | RelativeCS2 | | |
| | Position | 146 ,7.5 ,0 | um | 146um , 75um , 0um |
| | XSize | 64 | um | 64um |
| | YSize | -15 | um | -15um |
| | ZSize | 2 | um | 2um |

**Figure 3-44: Extension 2 Properties – *Command* Tab**

7. On the **Attribute** tab change the object **Name** to **Ext2** and click **OK**.
8. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

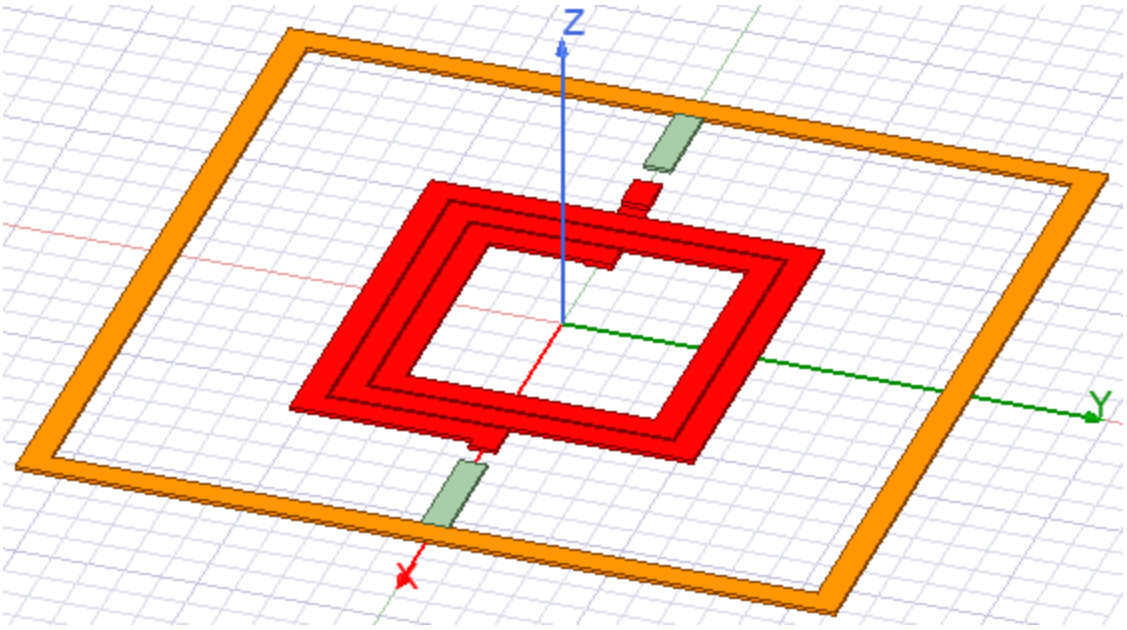



**Figure 3-45: Extension 2 Added to Ground Ring**

## Unite Ring Objects

You will now unite the ring and its extensions into a single object. This procedure completes the model geometry creation. Afterward, you can show the dielectric parts that were previously hidden.

1. Under *Model* > *Solids* > *pec* in the History Tree, select **GND_Ring, Ext1,** and **Ext2** (in that specific order).
2. On the **Draw** ribbon tab, click **Unite**.
3. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

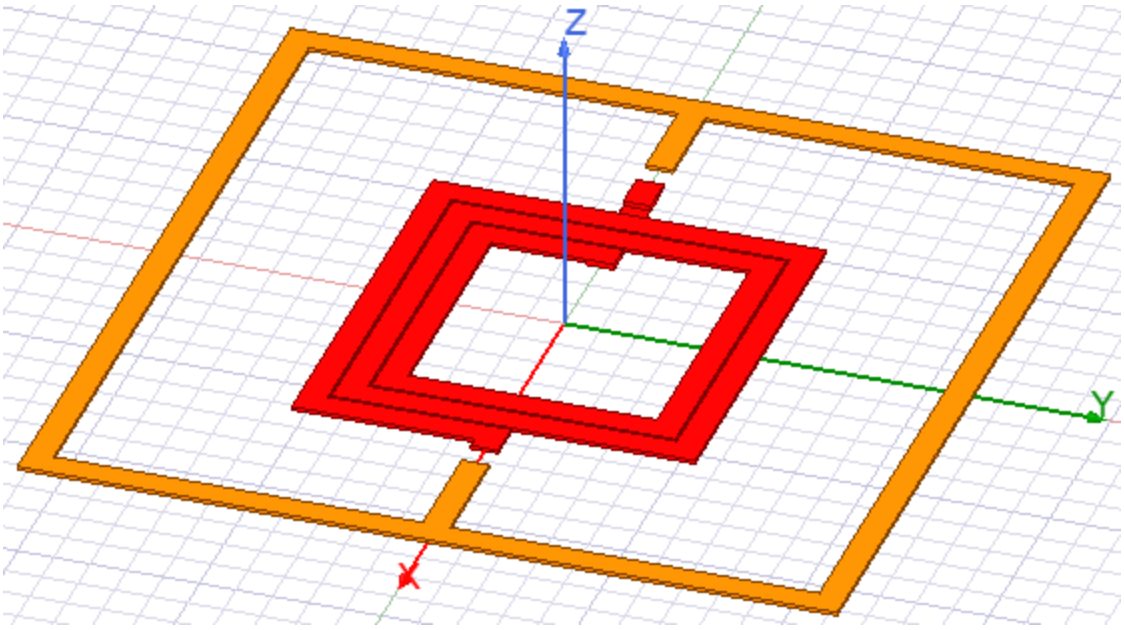



**Figure 3-46: Ground Ring and Extensions United**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 4 - Assign Boundaries and Excitations

You will now apply the boundary conditions and excitations needed to fully define the analysis model. Before applying the excitation, you must draw two rectangles to serve as the signal sources.

> **Note:**
>
> The order in which you assign boundaries can be important, as will be explained further in the topics that follow.

This subsection contains the following topics:

- Create Signal Sources
- Assign Excitation at Sources
- Assign Radiation Boundary
- Assign Perfect E Boundary to Ground Plane
- Boundary Display (Optional)

## Create Signal Sources

You will create signal sources at each end of the spiral inductor, at which excitations will be applied. The sources are drawn as rectangles, and each one extends from the centerline of one of the spiral inductor's end faces (termination) to the centerline of the adjacent ground ring extension's end face. The width of the source rectangles is the same as the spiral and extension width (15 µm).

To draw the rectangles, you can snap to the midpoint of a short edge at each termination face.

**Create *Source1* as follows:**

1. Zoom in for a close up view of the gap between the spiral's feed and the ground ring extension (the gap at the **-**X end of the spiral).
2. On the **Draw** ribbon tab, click **Draw rectangle**.
3. Click the following point to start the rectangle (a triangle appears to indicate the midpoint snapping point):


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

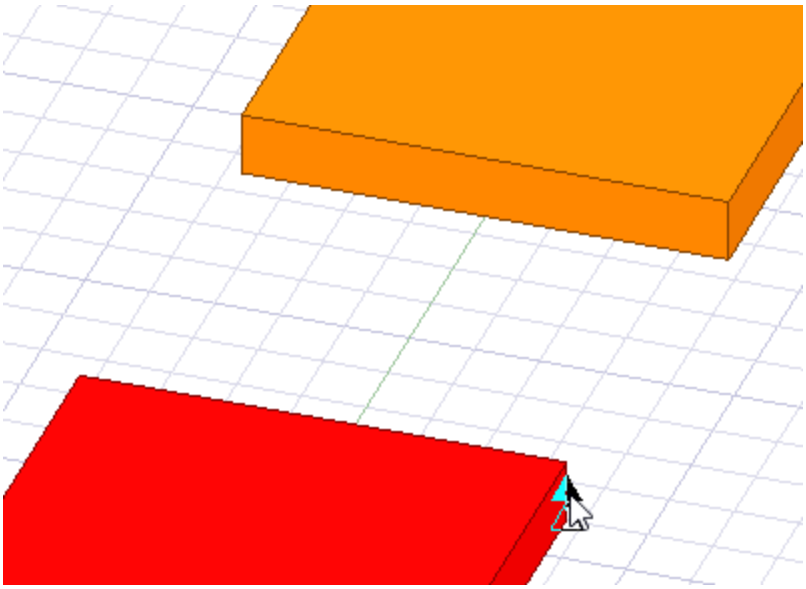


**Figure 4-1: First Corner of Source 1 Rectangle**

4. Click the following point to complete the rectangle:

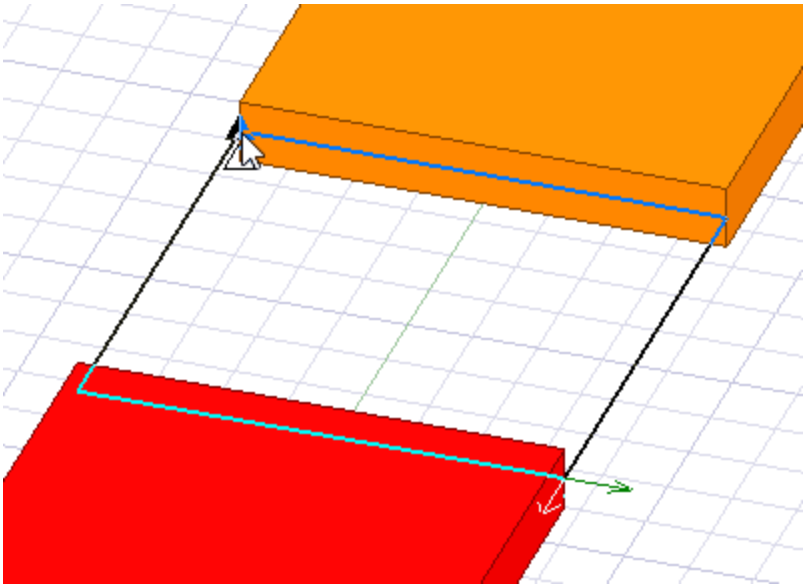


**Figure 4-2: Second Corner of Source 1 Rectangle**

The *Properties* dialog box appears after the second click.

5. On the **Attribute** tab, make the following changes:
   a. Change the **Name** to **Source1**.
   b. Ensure that the **Material Appearance** option is **not** selected.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Set the **Color** to **black** (column 1, row 6 of the *Basic colors* samples; Red: 0, Green: 0, Blue: 0).

d. Set the **Transparent** value to **0.4**.

6. Click **OK** to accept the settings and close the *Properties* dialog box.
7. Clear the selection.

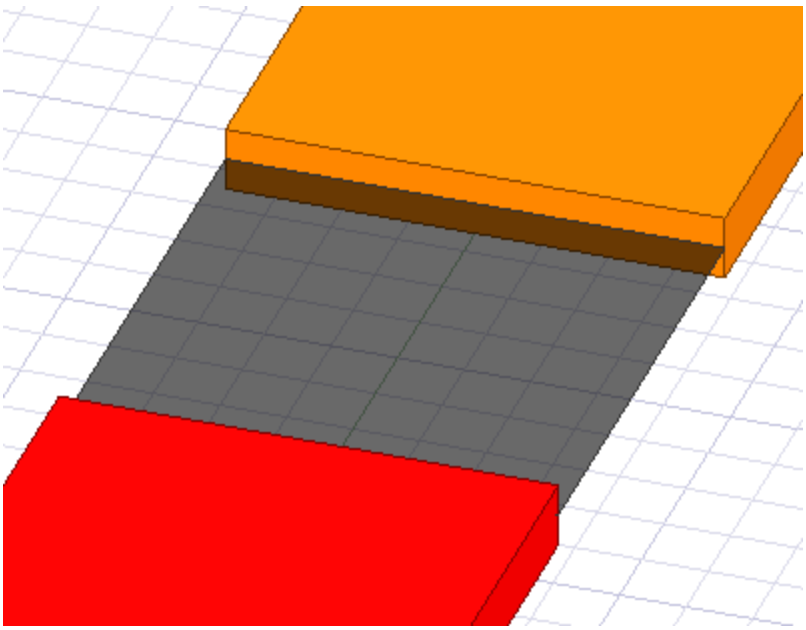


**Figure 4-3: Source 1 Drawn**

**Create *Source2* as follows:**

8. Repeat steps 1 through 10 but, this time:
   - Pan to, or zoom in on, the gap at the **+**X end of the spiral and draw the rectangle there.
   - Name the second rectangle **Source2**.

   When finished, the second source should look like the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

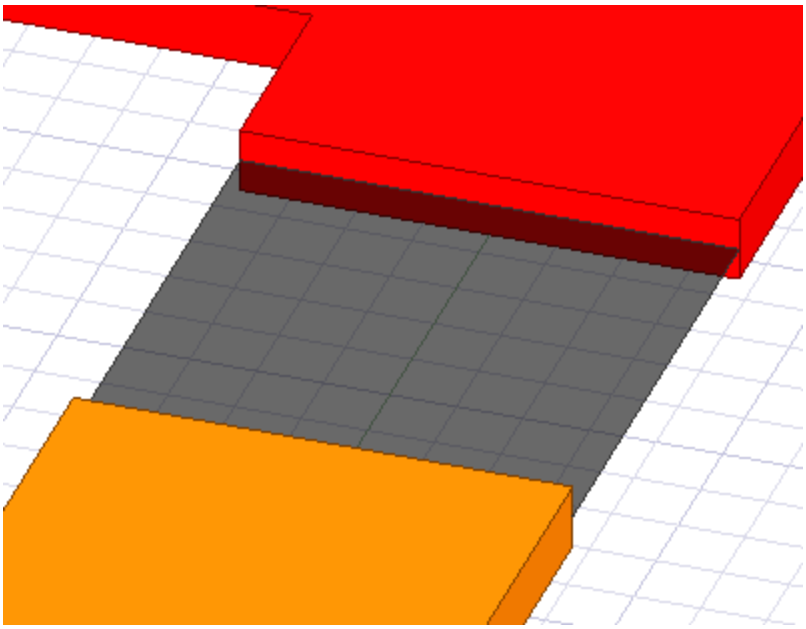


**Figure 4-4: Source 2 Drawn**

9. Press **Ctrl+D** to fit the view.

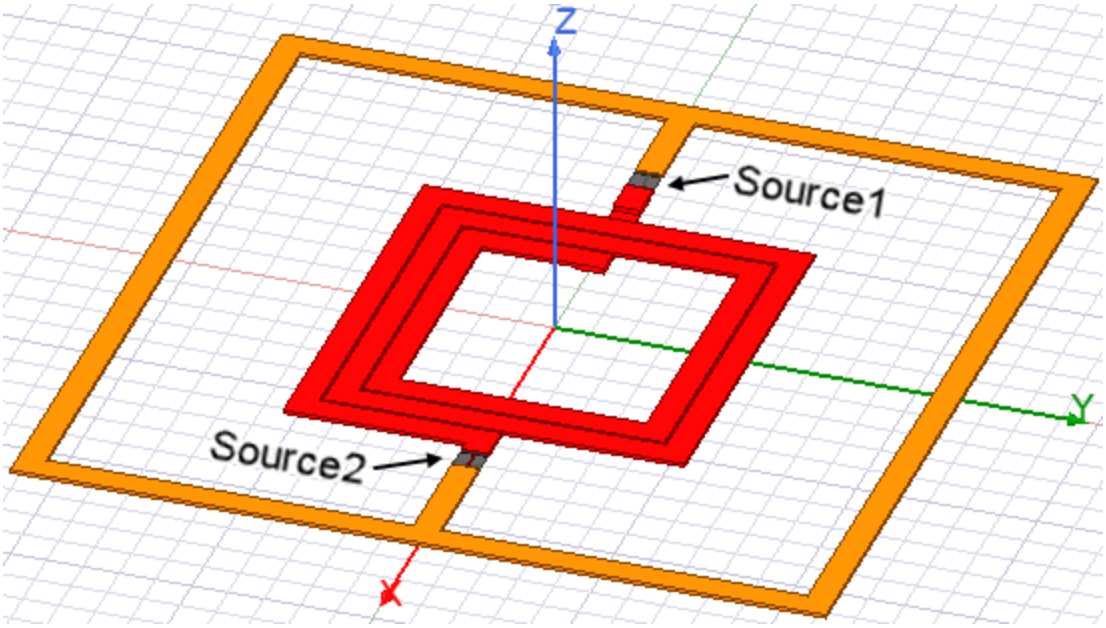



**Figure 4-5: *Source1* and *Source2* Added to Model**

# Assign Excitation at Sources

You will assign lumped port excitations at *Source1* and *Source2*.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

### Apply Lumped Port Excitation at Source1:

1. Under *Model* > *Sheets*> *Unassigned* in the History Tree, right-click **Source1** and choose **Assign Excitation** > **Port** > **Terminal Lumped Port** from the shortcut menu.

   The *Reference Conductors for Terminals* dialog box appears.

2. Define the port settings as shown in the following figure:

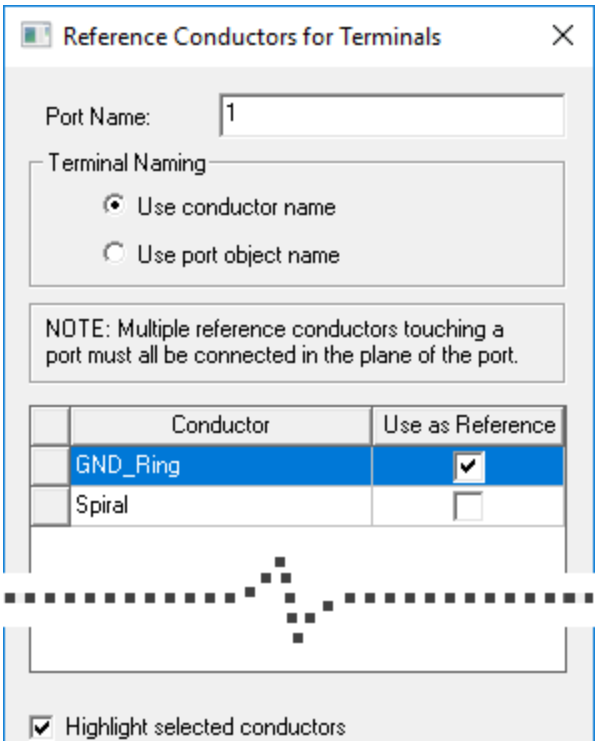



**Figure 4-6: Reference Conductor for Terminal – Port 1**

3. Click **OK** to apply the lumped port excitation.

### Apply Lumped Port Excitation at Source2:

4. Under *Model* > *Sheets* > *Unassigned* in the History Tree, right-click **Source2** and choose **Assign Excitation** > **Port** > **Terminal Lumped Port** from the shortcut menu.
5. In the *Reference Conductors for Terminals* dialog box that appears, specify the same settings as you did for the previous port (see *Port 1* figure), except for the **Port Name**, which is **2** this time.
6. Click **OK** to apply the lumped port excitation.

   The lumped ports (*1* and *2*) and their associated terminal definitions are listed under *Excitations* in the Project Manager:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

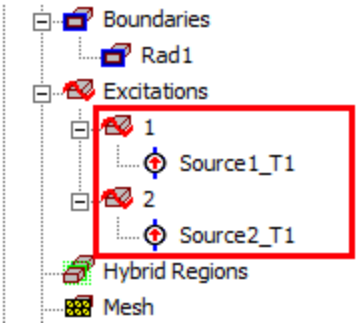



**Figure 4-7: Excitations in Project Manager**

With the definition of the excitations completed, you are done working on the conductors and can now show the hidden dielectric parts.

**To show all objects in the model, do the following:**

7. On the **Draw** ribbon tab, click **Show all objects in the active view**.
8. Do **Ctrl+D** to fit the view.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

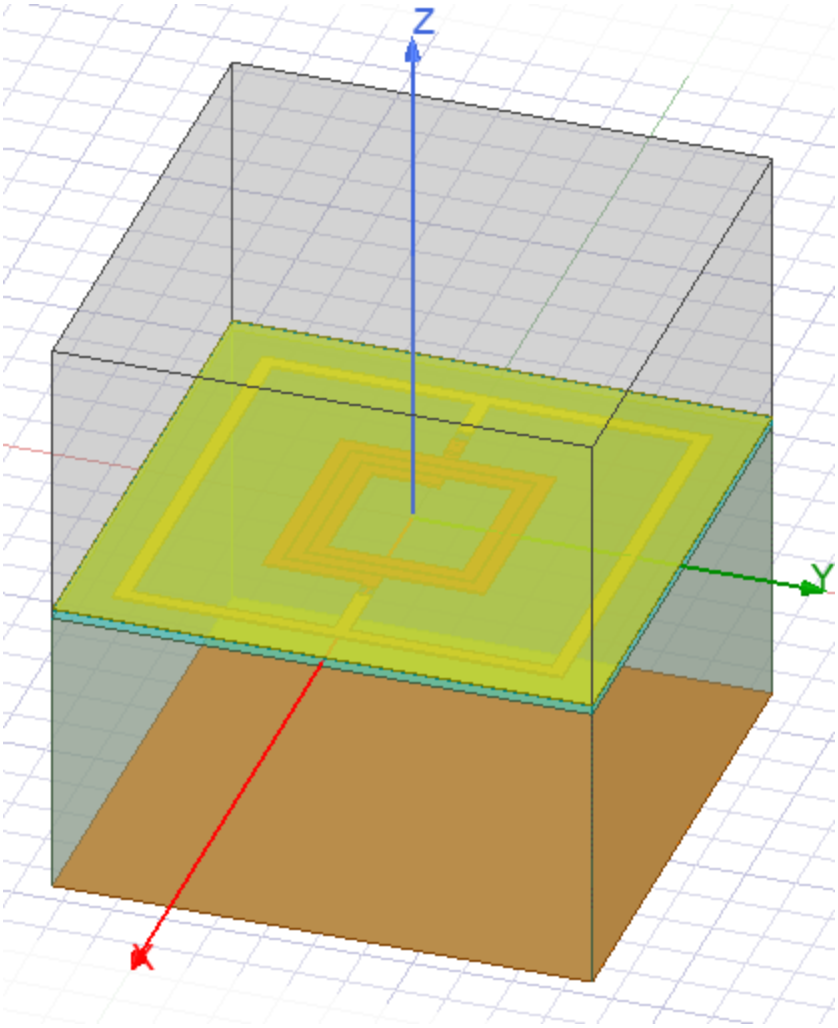



**Figure 4-8: All Objects Visible**

# Assign Radiation Boundary

Next, assign the radiation boundary to the air body, as follows:

1. Under *Model* > *Solids* > *air* in the History Tree, right-click **Air** and choose **Assign Boundary** > **Radiation** from the shortcut menu.

   The *Radiation Boundary* dialog box appears, which only has one setting.

2. Accept the default **Name** (**Rad1**) and click **OK**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

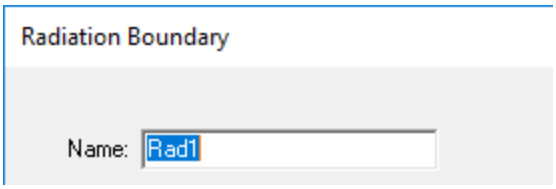



**Figure 4-9: Radiation Boundary Dialog Box**

3. If the radiation boundary visualization is not displayed on the model, select **Rad1** under *Boundaries* in the Project Manager to see it:

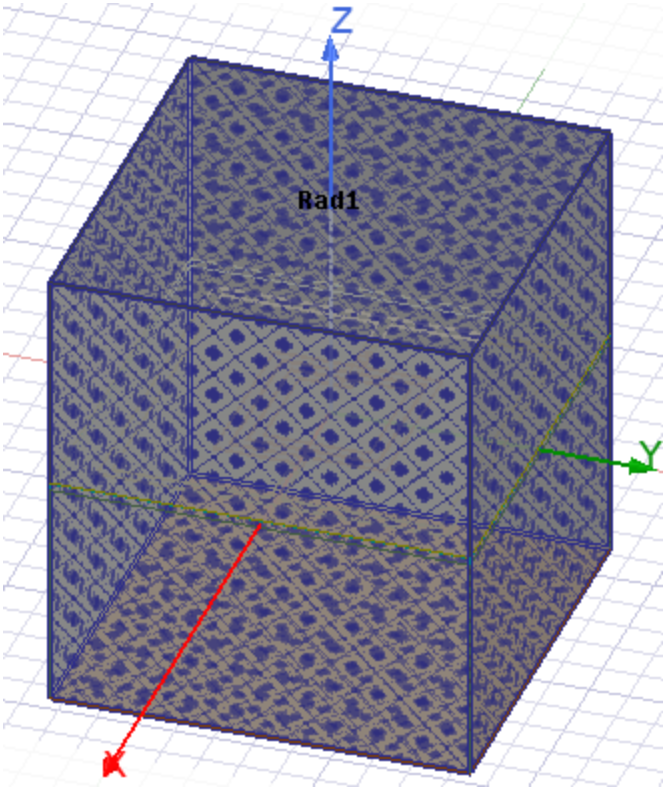



**Figure 4-10: Radiation Boundary Visualization**

4. Clear the selection.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Assign Perfect E Boundary to Ground

**Important:**

Because the face of the ground plane is coincident with the face of the air body, the order in which the radiation and perfect E boundaries are applied is important. Where the boundary conditions overlap, the latter application will override the earlier application. It is important that the ground plane behave like a perfect E boundary and not a radiation boundary. For this reason, you applied the radiation boundary before the perfect E boundary.

Assign the Perfect E boundary to the ground plane, as follows:

1. Under *Models* > *Sheets* > *Unassigned* in the History Tree, right-click **Ground** and choose **Assign Boundary** > **Perfect E** from the shortcut menu.

   The *Perfect E Boundary* dialog box appears, which has only two settings.

2. Ensure that the **Infinite Ground Plane** option is **not** selected, change the **Name** to **PerfE_Ground**, and click **OK**.

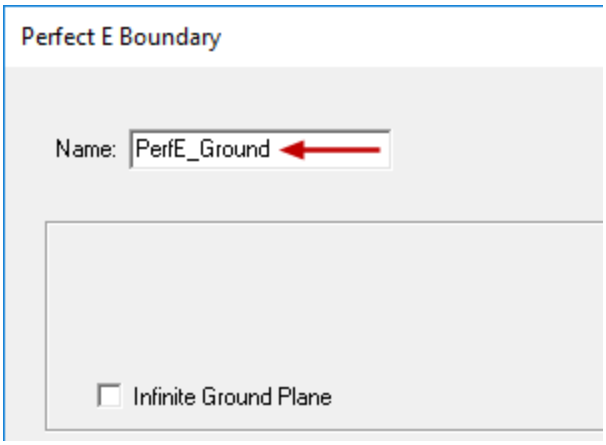



**Figure 4-11: *Perfect E Boundary* Dialog Box**

3. If the perfect E boundary visualization is not displayed on the model, select **PerfE_Ground** under *Boundaries* in the Project Manager to see it:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

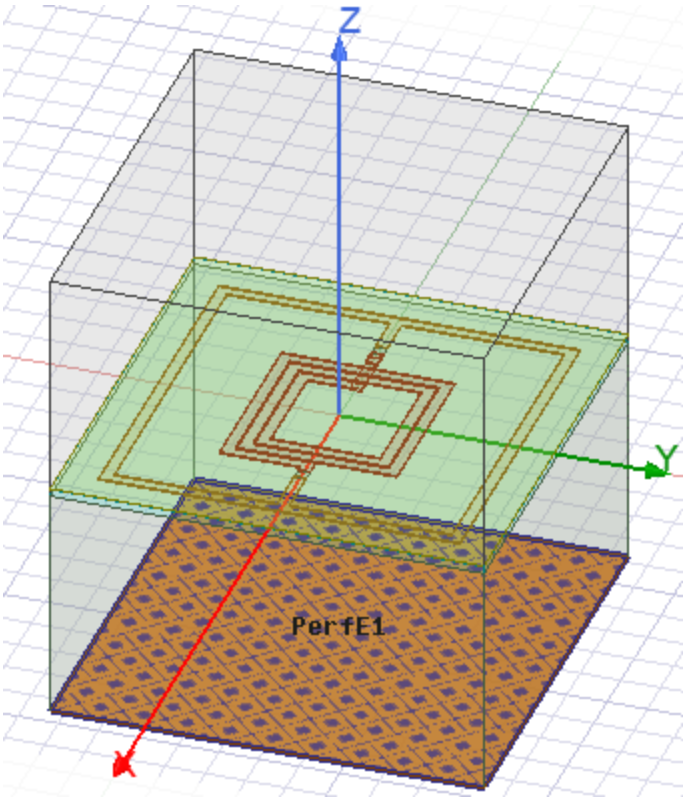



**Figure 4-12: Perfect E Boundary Visualization**

4. Clear the selection and **Save** your project.

# Boundary Display (Optional)

The solver view of boundaries provides a snapshot of all boundaries in the model (including ports and boundaries you've defined as well as default conditions applied to conductors and outer faces). It can be very useful for diagnosing problems with design setups. For example, you can see if a necessary boundary condition has been unintentionally superseded by one applied to a coincident face at a later time.

In the case of this example, you should verify that the perfect E boundary is in effect at the bottom face instead of the coincident radiation boundary.

The solver view of boundaries works best when the model rendering mode is Wireframe, instead of Smooth Shaded. In this way, only the boundary colors are rendered on model faces, and not the material appearance colors.

1. Rotate the model viewpoint so that you are looking upward at it from below, and the bottom face is not hidden.
2. Using the menu bar, click **Render** > **Wireframe** (or press **F6**).


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now, only the edges of objects are visible.

3. Using the menu bar, click **HFSS** > **Boundary Display (Solver View)**.

   The *Solver View of Boundaries* dialog box appears.

> **Note:**
>
> HFSS identifies all the unique boundary conditions and ports that exist in the model (user-applied and default).

4. Click the **Visibility** column heading to turn on the display of all boundaries in a single click.

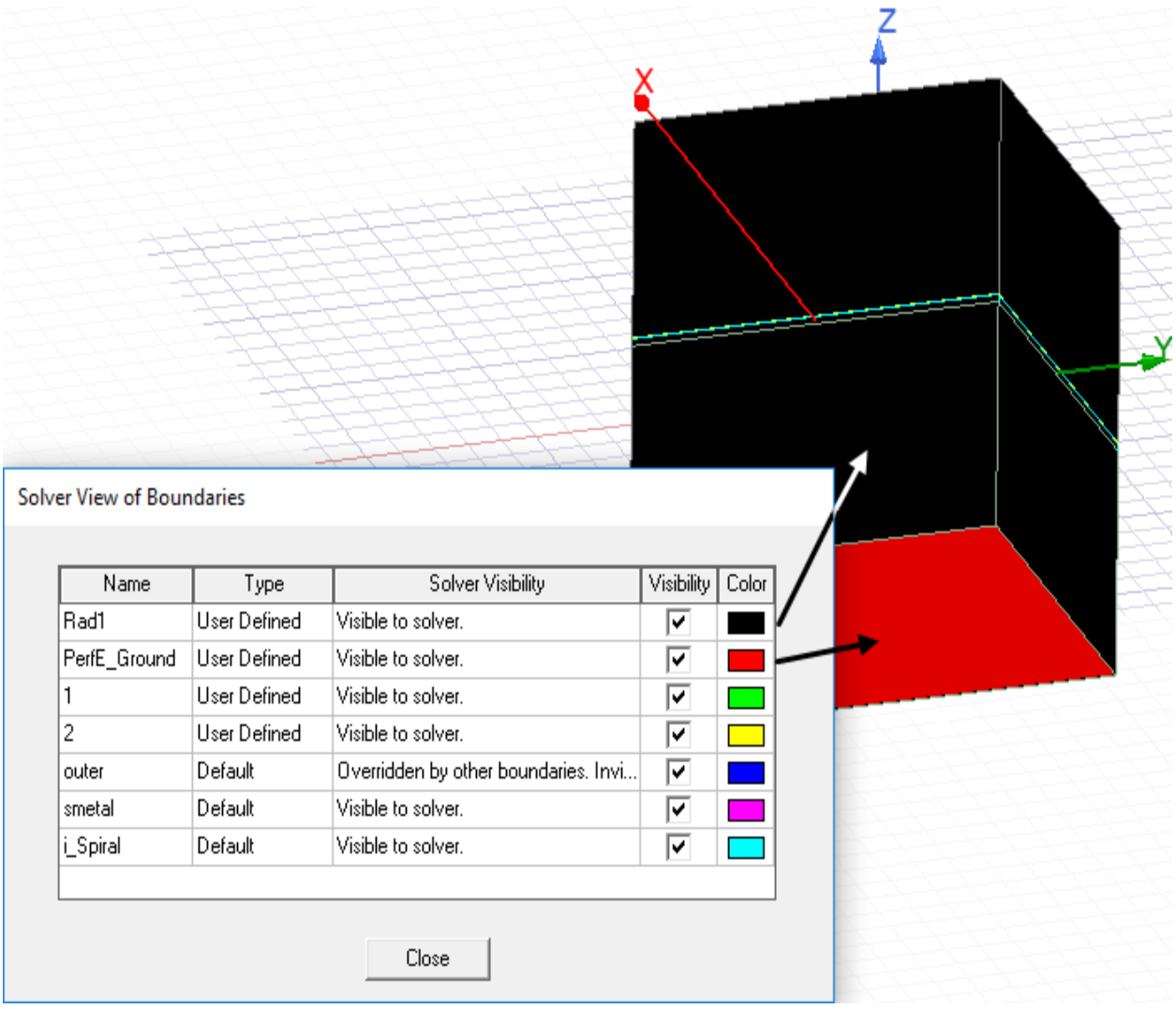



**Figure 4-13: Solver View of All Boundaries**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notice that the *PerfE_Ground* boundary condition overrode the *Rad1* boundary that was applied to all faces of the air body. This is the correct result for this model.

5. Deselect the *Visibility* option for the **PerfE_Ground** boundary.

   Notice that the color of the bottom edges is now orange, which is the ground plane object's color (that is, material appearance). This color is not associated with any of the boundary conditions or ports. Therefore, the solver only sees the *PerfE_Ground* boundary, which was applied after the *Rad1* boundary and overrode it.

6. Deselect the *Visibility* option for the **Rad1** boundary.
7. On the **Draw** ribbon tab, click **Orient** to restore the default *Trimetric* model viewpoint. (You do not have to access the *Orient* drop-down menu.)
8. Zoom in and pan the model so that you can clearly see all of the boundary colors on the spiral inductor, ground ring, and sources:

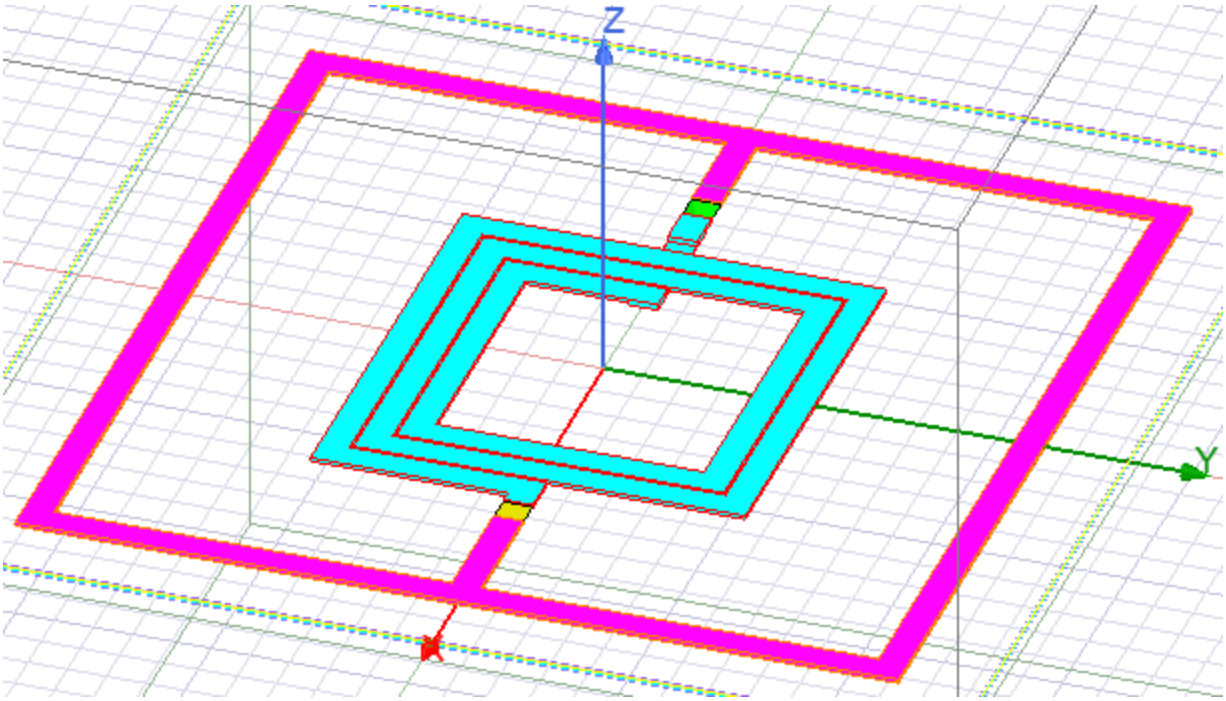



**Figure 4-14: Solver View or Conductors and Ports**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you double-click any swatch in the **Color** column, you can change the color as desired from the palette that appears. However, the overrides are only in effect while the dialog box is open. If you close it and choose the *Boundary Display (Solver View)* command again, the colors will revert to their default settings.

9. Click **Close**.
10. From the menu bar, click **Render** > **Smooth Shaded** (or press **F7**) to restore the shaded model display.
11. Press **Ctrl+D** to fit the view.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 5 - Analyze the Spiral Inductor

This chapter describes how to setup and run the simulation (including assigning mesh refinement). It also covers reviewing the solution data, and generating reports.

This chapter contains the following topics:

- Create Analysis Setup
- Add Frequency Sweep
- Model Validation
- Analyze the Model
- Review the Solution Data
    - Profile
    - Convergence
    - Matrix Data
    - Mesh Statistics
- Create S-Parameter vs. Frequency Plot
- Custom Equations – Output Variables
- Simulate with *Solve Inside Conductors*
- Results with *Solve Inside Conductors*

## Create Analysis Setup

To create an analysis setup:

1. On the **Simulation** ribbon tab, click **Setup** > **Advanced**.

   The *Driven Solution Setup* dialog box appears.

2. In the **General** tab, edit the settings as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

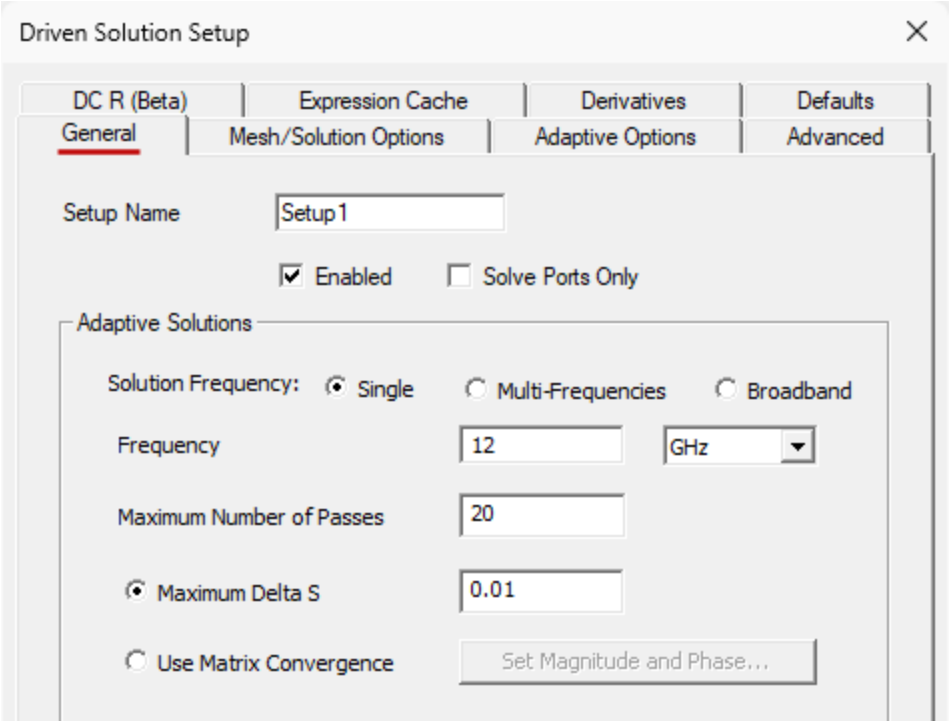



**Figure 5-1: Driven Solution Setup – *General* Tab**

3. In the **Mesh/Solution Options** tab, edit the settings as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

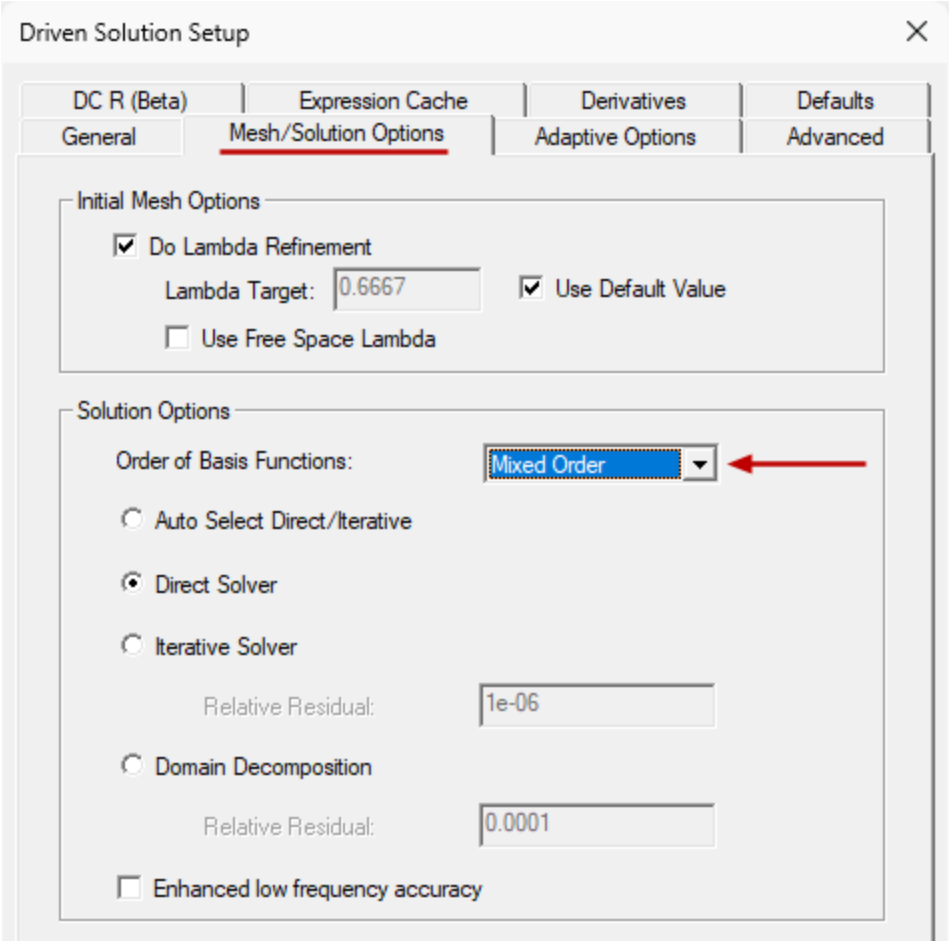



**Figure 5-2: Driven Solution Setup – *Mesh/Solution Options* Tab**

4. Click **OK**.

   Because at least one port was defined prior to completing the solution setup, the *Edit Frequency Sweep* dialog box opens automatically. Keep this dialog box open and proceed to the next page, where you will find the instructions for defining the frequency sweep.

# Add a Frequency Sweep

The *Edit Frequency Sweep* dialog box should already be open. However, if you accidentally closed it, reopen the dialog box by completing steps 1 and 2. Otherwise, skip to step 3:

1. Under *Analysis* in the Project Manager, select **Setup1**.
2. On the **Simulation** ribbon tab, click **Sweep** (Add Frequency Sweep).

   The *Edit Frequency Sweep* dialog box appears.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define the sweep settings as follows:

3. In the **General** tab, specify the following settings:

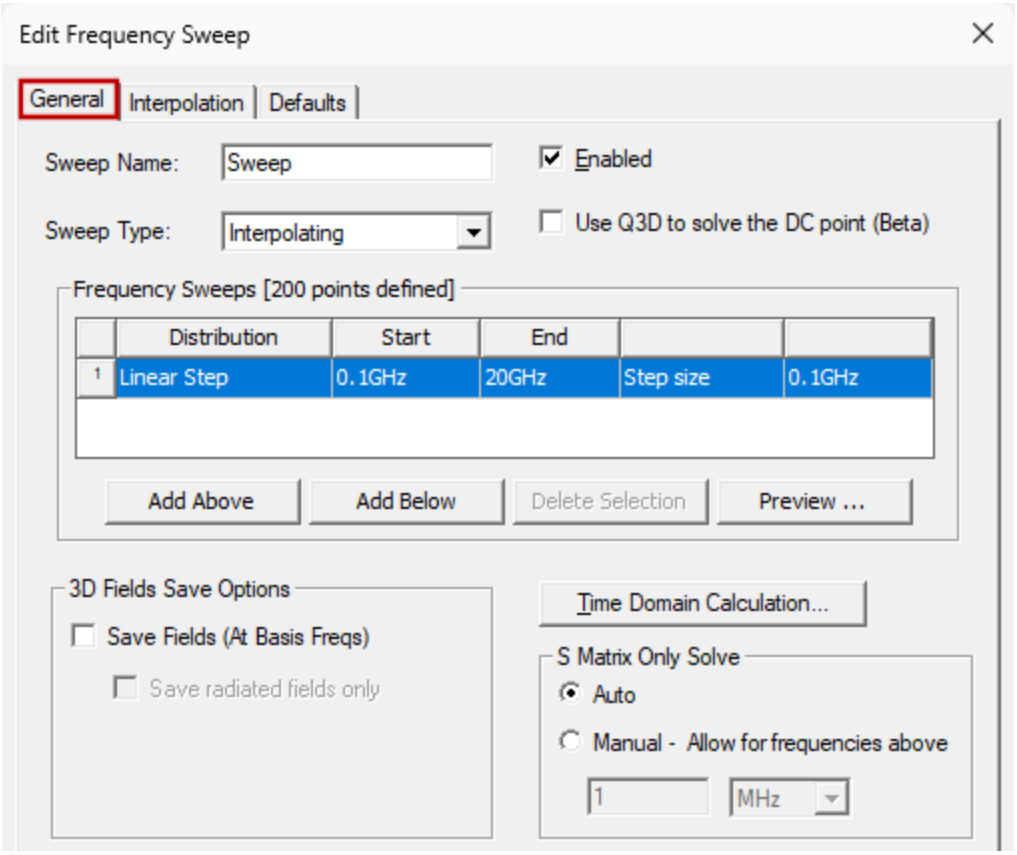



**Figure 5-3: Edit Frequency Sweep – *General* Tab**

4. In the **Interpolation** tab, edit the settings as in shown in the following figure:

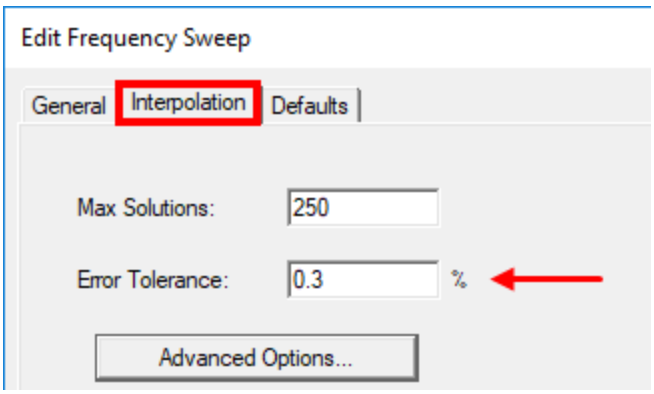



**Figure 5-4: Edit Frequency Sweep – *Interpolation* Tab**

5. Click **OK**.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Assign Mesh Refinement

In this section you will set HFSS to refine the length of the tetrahedral elements for the spiral until they are below the specified value.

1. Under *Model* > *Solids* > *My_Metal* in the History Tree, right-click **Spiral** and choose **Assign Mesh Operation** > **Inside Selection** > **Length Based**.

   The *Element Length Based Refinement* dialog box appears.

2. Edit the settings as shown in the figure below and click **OK**.

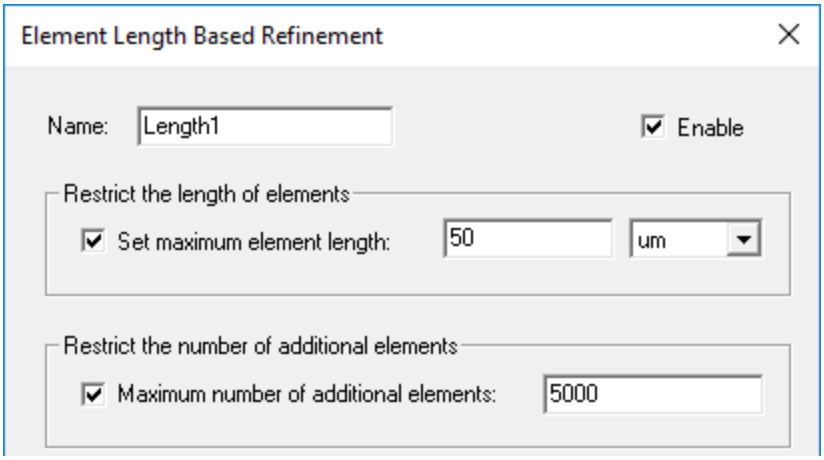



**Figure 5-5: Element Length Based Refinement Settings**

3. Under *Mesh* in the Project Manager, right-click **Length1** and click **Select Assignment** to highlight the object to which the mesh operation is assigned.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

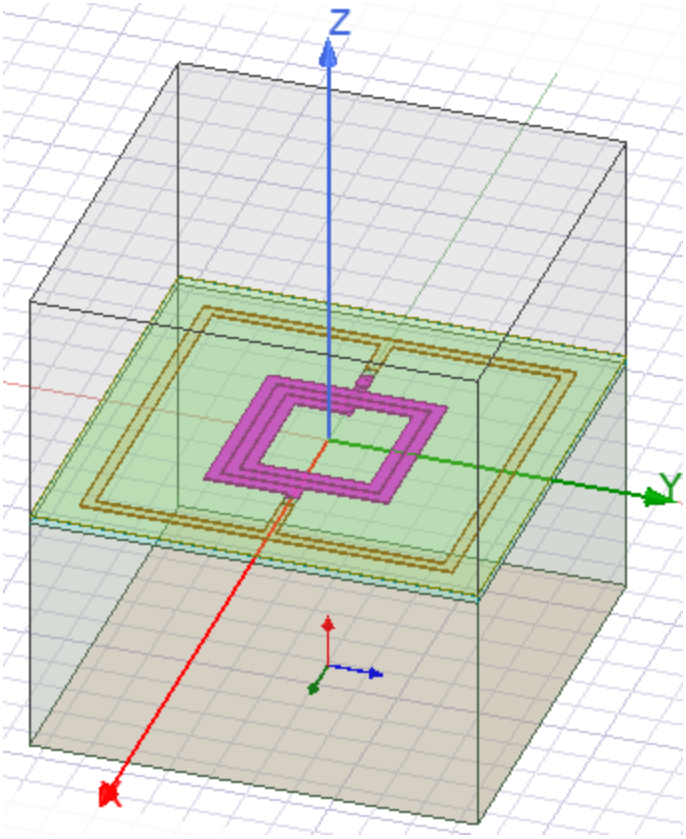



**Figure 5-6: Identifying Target Object for Mesh Operation**

4. Click in the Modeler window's background area to clear the selection.

# Validate and Analyze

Before running the simulation your model must pass the **Validation Check**.

**To validate the model:**

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

   The *Validation Check* window appears and the model is validated:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

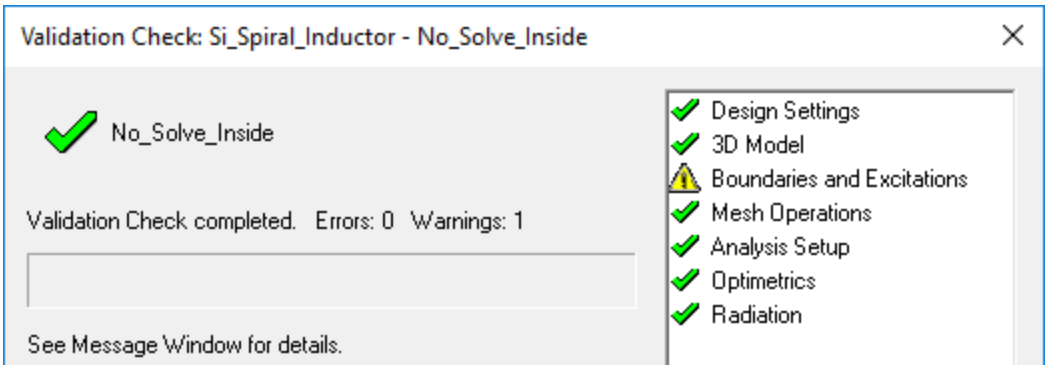



**Figure 5-7: Validation Check**

> **Note:**
>
> The *Message Manager* window gives the details concerning the *Boundaries and Excitations* warning. Specifically, the warning says that, *Boundary "Rad1" and Boundary "PerfE_Ground" overlap*. You can disregard this warning because you've already verified that the solver sees the correct boundary condition where they overlap (at the bottom face of the model). To refresh your memory, return to the Boundary Display topic. When applying the radiation boundary, you could have selected five of the six air body faces, thus avoiding the overlap and the warning message. But it is not necessary for you to do so, since the latter condition (*PerfectE_Ground*) overrides the former condition (*Rad1*). Applying the boundary condition to the whole object was simply more convenient.

2. If you see any other warnings or errors (aside from the overlapping *PerfE_Ground* and *Rad1* boundaries), recheck your steps in building and setting up the model.
3. Click **Close**.

**To start the solution process:**

4. On the **Simulation** ribbon tab, click **Analyze All**.

   The solution will take a few minutes to complete, with the time depending on your computer hardware.

# Review Solution Data

You can review the solution data while the simulation is running. For example, you can watch the adaptive passes in the *Convergence* tab and see if the solution is trending toward convergence


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or is diverging. Of course, you won't see the a complete log of the solution data until the analysis is finished.

The solution data is presented in the Solutions dialog box, which consists of the following four tabs. Each tab is covered in a separate subtopic:

- Profile
- Convergence
- Matrix Data
- Mesh Statistics

There are several ways to access the solution data:

- On the **Results** ribbon tab, click **Solution Data**. Then, select the desired tab in the *Solutions* dialog box.
- Right-click **Results** in the Project Manager and choose **Solution Data** from the shortcut menu. Then, select the desired tab in the *Solutions* dialog box.
- Right-click **Setup1** (under Analysis in the Project Manager) and directly select the solution data tab you want to display (**Profile, Convergence, Matrix Data,** or **Mesh Statistics**).
- Using the menu bar, click **HFSS** > **Results** > **Solution Data**. Then, select the desired tab in the *Solutions* dialog box.

# Review the Profile Panel

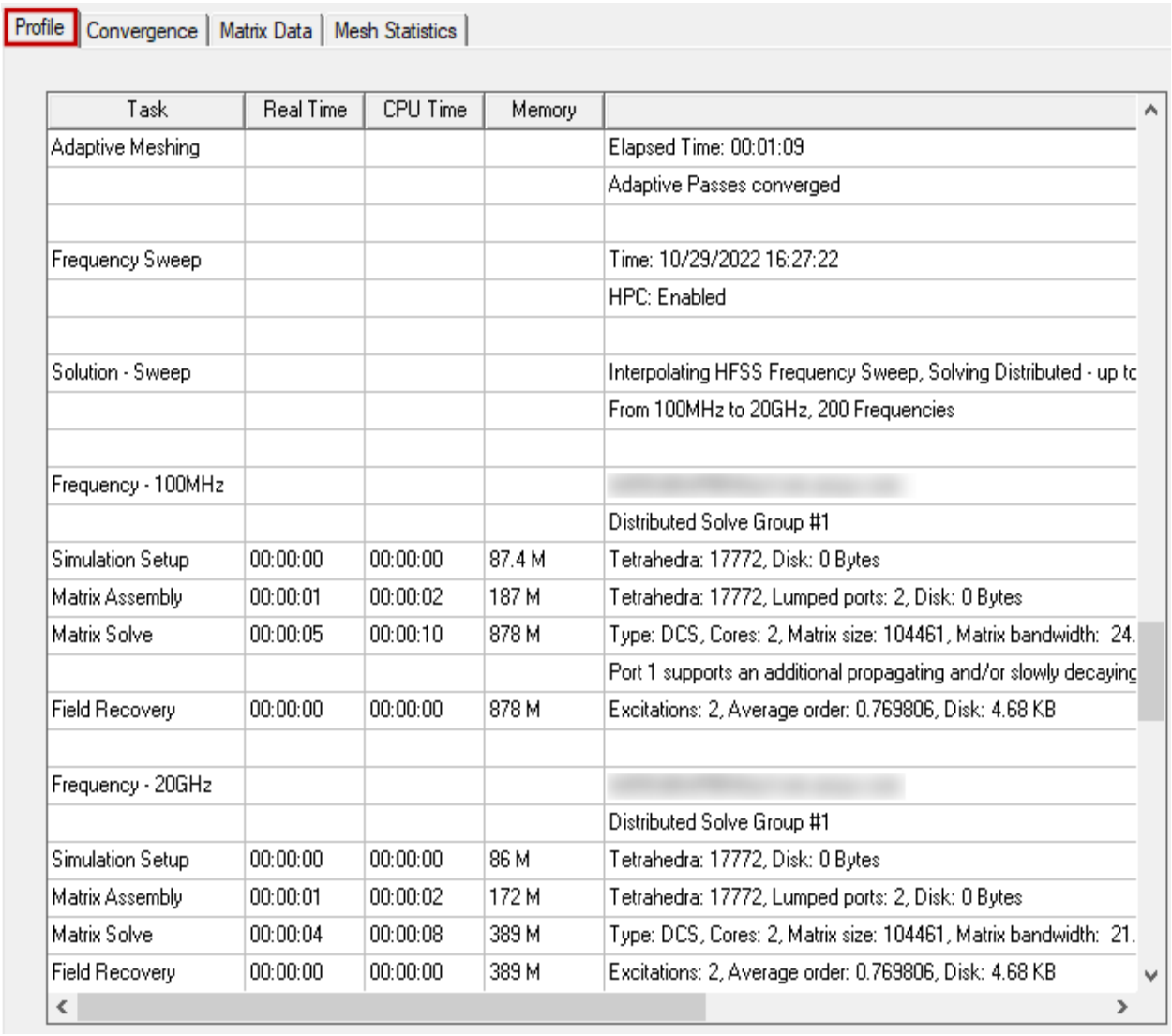
The Profile window shows you a synopsis of the simulation process, providing real time, CPU time, and memory consumed for each phase of the process. The log includes mesh creation and refinement, port adaptation, adaptive passes, matrix assembly (S, Y, and Z coefficients), solver, field recovery, data transfer, frequency sweep, and more. The total real time and CPU time for the entire solution process are also given. The more highly refined the mesh (that is, the higher the number of tetrahedra generated), the more accurate the solution will be. Optimal results are achieved by only refining the mesh adaptively where required (specifically, in the areas of greatest solution error). There is a trade-off between the number of tetrahedra used and the computational resources required. Increased accuracy requires more computational resources and more time to solve.

1. Under *Analysis* in the Project Manager, right-click **Setup1** and choose **Profile** from the shortcut menu.

   The *Profile* tab of the *Solutions* dialog box appears.

2. Review the profile data.
3. Keep the dialog box open and proceed to the next topic.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.




| Task | Real Time | CPU Time | Memory | |
|---|---|---|---|---|
| Adaptive Meshing | | | | Elapsed Time: 00:01:09 |
| | | | | Adaptive Passes converged |
| | | | | |
| Frequency Sweep | | | | Time: 10/29/2022 16:27:22 |
| | | | | HPC: Enabled |
| | | | | |
| Solution - Sweep | | | | Interpolating HFSS Frequency Sweep, Solving Distributed - up to |
| | | | | From 100MHz to 20GHz, 200 Frequencies |
| | | | | |
| Frequency - 100MHz | | | | [redacted] |
| | | | | Distributed Solve Group #1 |
| Simulation Setup | 00:00:00 | 00:00:00 | 87.4 M | Tetrahedra: 17772, Disk: 0 Bytes |
| Matrix Assembly | 00:00:01 | 00:00:02 | 187 M | Tetrahedra: 17772, Lumped ports: 2, Disk: 0 Bytes |
| Matrix Solve | 00:00:05 | 00:00:10 | 878 M | Type: DCS, Cores: 2, Matrix size: 104461, Matrix bandwidth: 24. |
| | | | | Port 1 supports an additional propagating and/or slowly decaying |
| Field Recovery | 00:00:00 | 00:00:00 | 878 M | Excitations: 2, Average order: 0.769806, Disk: 4.68 KB |
| | | | | |
| Frequency - 20GHz | | | | [redacted] |
| | | | | Distributed Solve Group #1 |
| Simulation Setup | 00:00:00 | 00:00:00 | 86 M | Tetrahedra: 17772, Disk: 0 Bytes |
| Matrix Assembly | 00:00:01 | 00:00:02 | 172 M | Tetrahedra: 17772, Lumped ports: 2, Disk: 0 Bytes |
| Matrix Solve | 00:00:04 | 00:00:08 | 389 M | Type: DCS, Cores: 2, Matrix size: 104461, Matrix bandwidth: 21. |
| Field Recovery | 00:00:00 | 00:00:00 | 389 M | Excitations: 2, Average order: 0.769806, Disk: 4.68 KB |

**Figure 5-8: *Solutions* Dialog Box – *Profile* Tab**

# Review the Convergence Panel

1. Select the **Convergence** tab of the *Solutions* dialog box.

   The default *View* option is *Table*.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

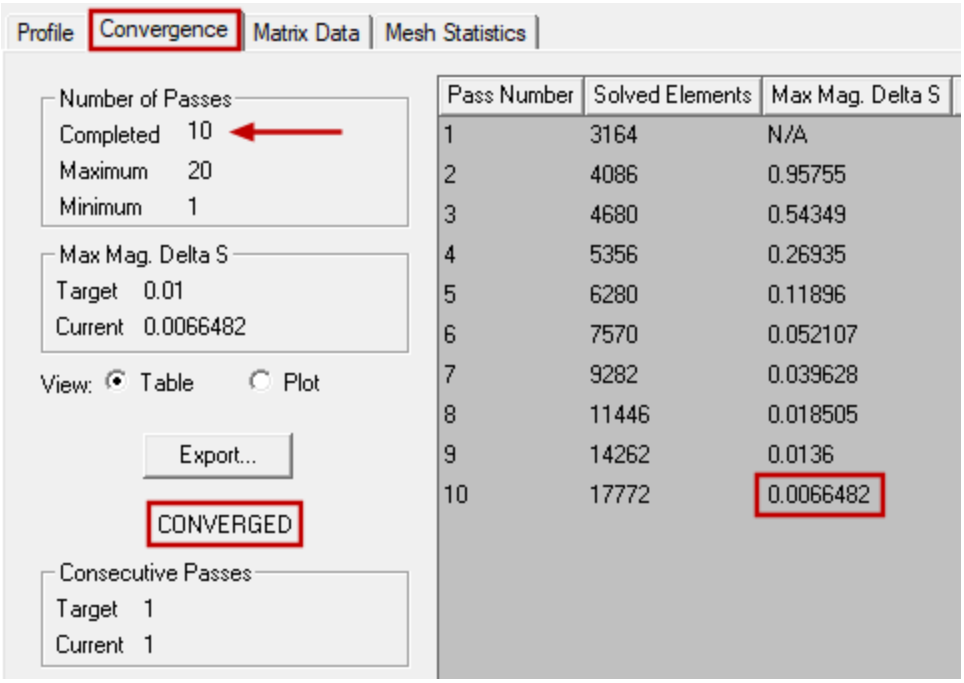



**Figure 5-9: Convergence Table**

> **Note:**
>
> This solution is converged (that is, a Max. Mag. Delta S result of less than or equal to the *Target* value of 0.01 was achieved between consecutive adaptive passes).

2. Select the **Plot** option to view a graphical representations of the convergence data:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

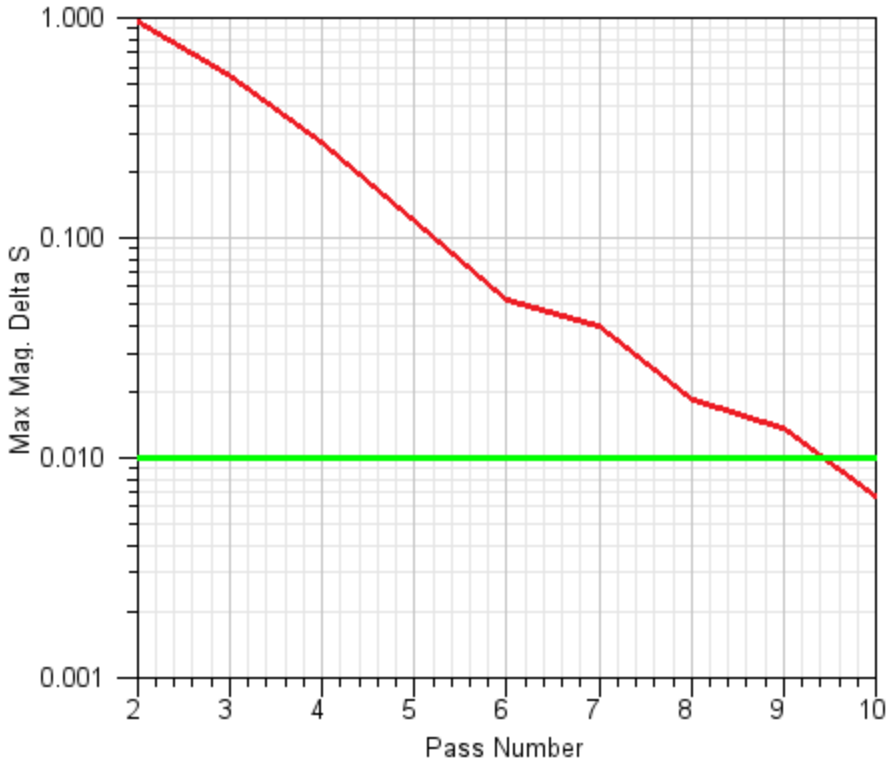



**Figure 5-10: Convergence Plot**

3. Keep the dialog box open and proceed to the next topic.

# Review the Matrix Data Panel

In this tab, you can view the computed S, Y, Z, Gamma (when applicable), or Zo matrix coefficients (representing scattering, admittance, impedance, propagation, and terminal characteristic impedance, respectively). You can view this data for the adaptive passes, the last adaptive pass, or the frequency sweep using the drop-down menus at the top of the dialog box. Additionally, you can view the results at specific frequencies or display the results for all computed frequencies.

1. Select the **Matrix Data** tab.
2. From the **Simulation** drop-down menus, select **Setup1** and **Sweep**.

   The S matrix for the first frequency in the sweep is shown by default. Other frequencies are selectable via the drop-down menu in the *View* subtab:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

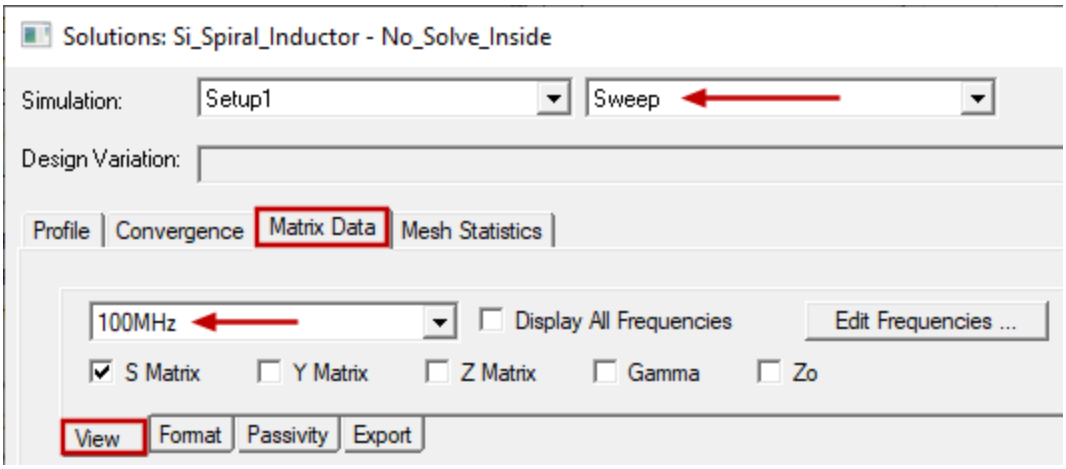



**Figure 5-11: Matrix Data – *View* Options**

3. Select the **Display All Frequencies** option:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

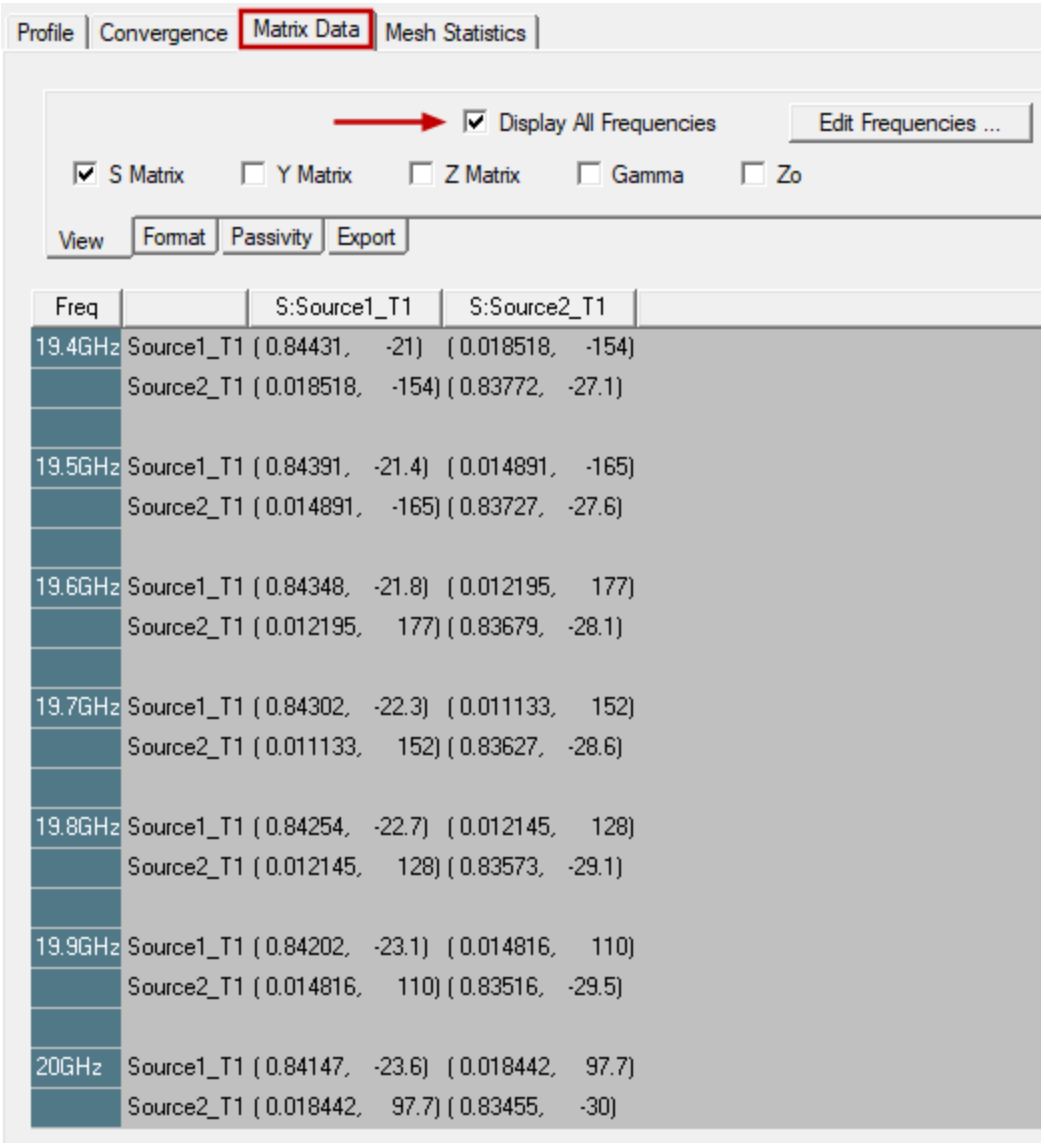



**Figure 5-12: Matrix Data**

> **Note:**
>
> To view a real-time update of the matrix data while a solution is still being solved, set the **Simulation** options to **Setup1** and **Last Adaptive** using the drop-down menus at the top of the dialog box.

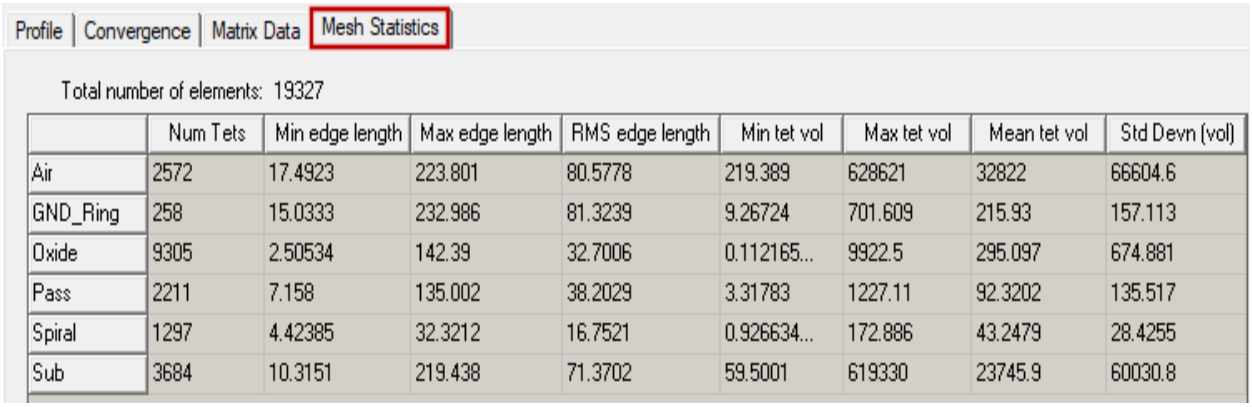
4. Keep the dialog box open and proceed to the next topic.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Review the Mesh Statistics Panel

As the title indicates this panel shows statistics about the mesh that was generated. More specifically, it gives a breakdown of the statistics, tabulating the data for each individual solid object comprising the model. The data includes the number of tetrahedra generated; the minimum, maximum , and RMS (root mean squared) edge length; the minimum, maximum, and mean tetrahedron volume; and the standard deviation of the tetrahedra volumes.

1. Select the **Mesh Statistics** tab:



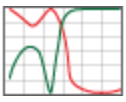

| | Num Tets | Min edge length | Max edge length | RMS edge length | Min tet vol | Max tet vol | Mean tet vol | Std Devn (vol) |
|---|---|---|---|---|---|---|---|---|
| Air | 2572 | 17.4923 | 223.801 | 80.5778 | 219.389 | 628621 | 32822 | 66604.6 |
| GND_Ring | 258 | 15.0333 | 232.986 | 81.3239 | 9.26724 | 701.609 | 215.93 | 157.113 |
| Oxide | 9305 | 2.50534 | 142.39 | 32.7006 | 0.112165... | 9922.5 | 295.097 | 674.881 |
| Pass | 2211 | 7.158 | 135.002 | 38.2029 | 3.31783 | 1227.11 | 92.3202 | 135.517 |
| Spiral | 1297 | 4.42385 | 32.3212 | 16.7521 | 0.926634... | 172.886 | 43.2479 | 28.4255 |
| Sub | 3684 | 10.3151 | 219.438 | 71.3702 | 59.5001 | 619330 | 23745.9 | 60030.8 |

**Figure 5-13: Mesh Statistics**

Notice that the Spiral object has the number of tetrahedral element reported along with their length and volume data. A solid mesh is generated for all objects, whether each mesh is used or not. For this analysis, the *Spiral* object's *Solve Inside* option was deselected. So, this particular mesh was not actually used in the solution.

2. Click **Close** when you are done reviewing the contents of this tab.

# Create S-Parameter vs. Frequency Plot

1. On the Results ribbon tab, click **Terminal Solution Data Report** >  **2D**.

   The *Report* dialog box appears.

2. Edit the settings as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

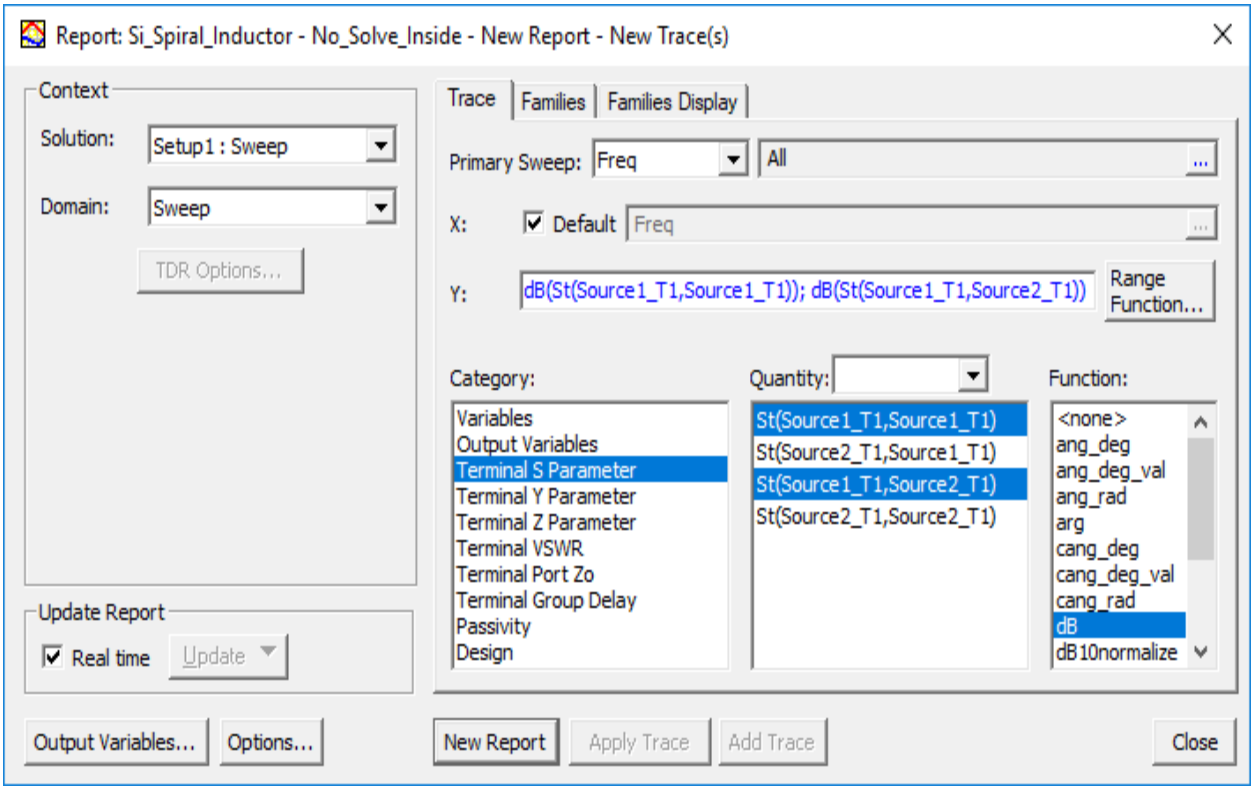



**Figure 5-14: S Parameter vs. Frequency Plot Settings**

3. Click **New Report** but keep the dialog box open for now.

   The Terminal S Parameter plot appears in a new window:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

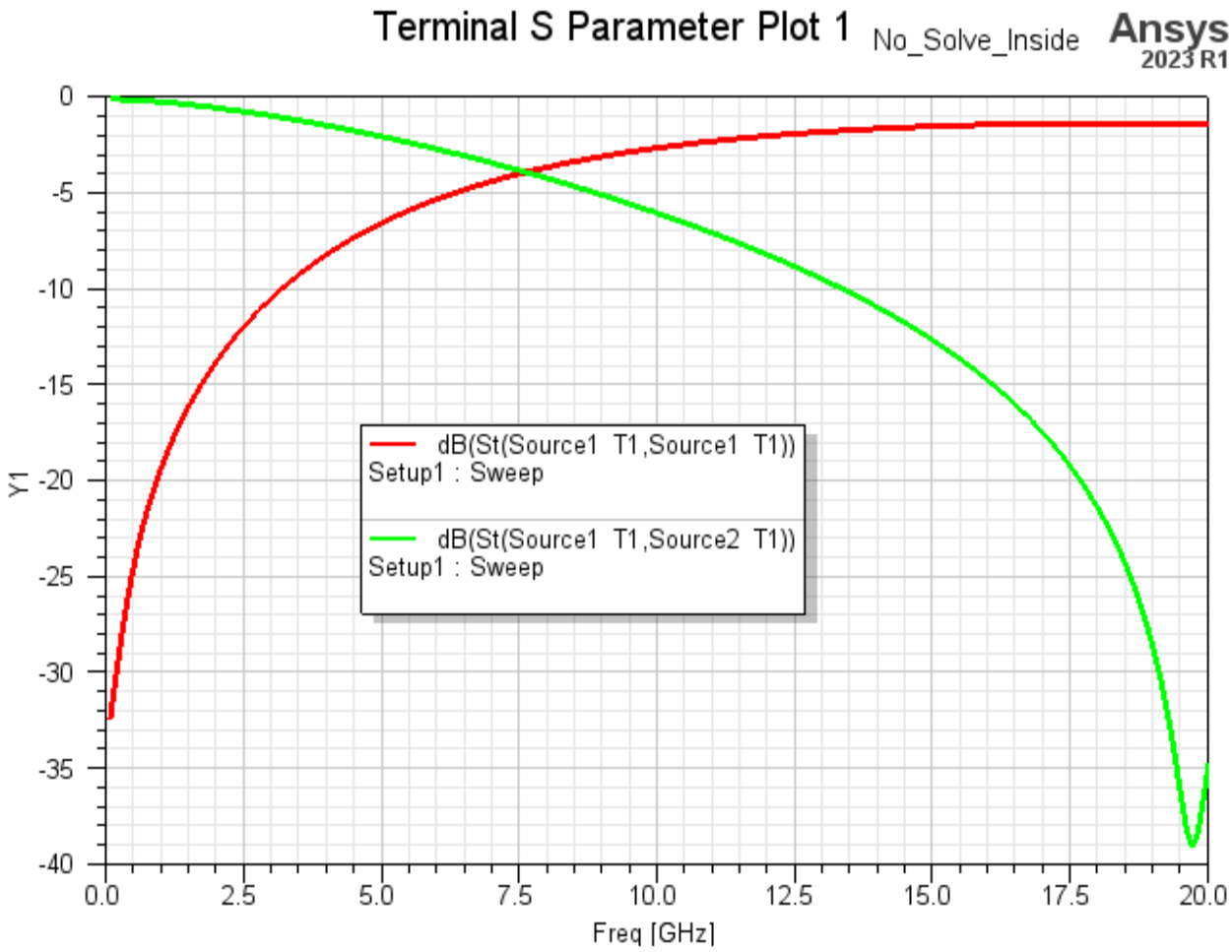



**Figure 5-15: S-Parameter vs. Frequency Plot (No_Solve_Inside)**

Notice that the basic trend of the curves is as you would expect for an inductor. As the frequency increases, the transmission to the second terminal decreases (green curve) and the signal reflection at the input terminal increases. Inductors allow direct current to pass unimpeded but increasingly attenuate the signal as the frequency increases.

# Custom Equations – Output Variables

In this procedure, you will create two plots based on user-defined output variables. You will define three output variables, *R, L,* and *Q*, and plot each of these variables versus frequency.

A real-world inductor includes the inductance (L), a parasitic series resistance (R), and parasitic capacitance (C) in the substrate between the inductor and ground plane. The following figure is a representative "pi" circuit for modeling a real-world spiral inductor:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

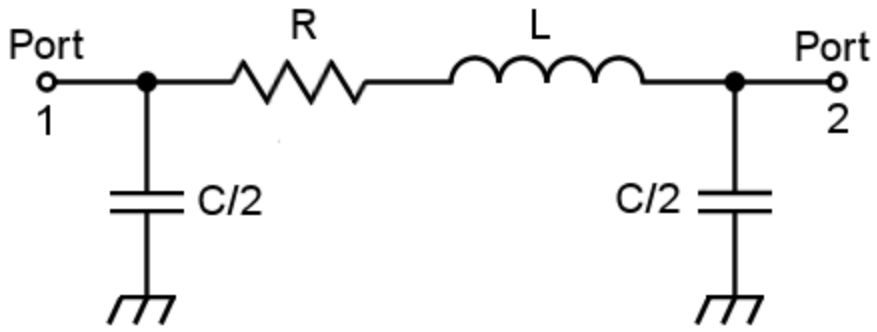



**Figure 5-16: Simple "Pi" Model of a Spiral Inductor**

The structure is assumed to be roughly symmetrical. So, as an approximation, the capacitance is divided into two lumped capacitors, each with a value C/2, and each attached to one of the two ports. For simplicity, assume that the R, L, and C values are all independent of frequency. However, in reality, the resistance in particular will increase with frequency due to the skin effect. Additionally, in order to easily extract values from the HFSS results, assume that the circuit is working at a frequency well below its resonant frequency. Finally, assume that the product of the parasitic resistance and capacitance is sufficiently low. That is, assume $\omega R(C/2) \approx 0$, where $\omega$ is the frequency in radians/second.

For the purpose of this exercise, you will not extract or plot R or C values. However, the R value is a constituent of the quality factor (Q), which will be the subject of your second output variable and plot.

## Inductance (*L*) vs. Frequency:

Based on the previously stated assumptions, it is possible to extract the approximate L value of the inductor from the Y-parameter results produced by the HFSS analysis (specifically, $Y_{11}$). The equation is as follows:

$L \approx im(1/Y_{11})/\omega = im(1/Y_{11}) / (2 \cdot \pi \cdot Freq)$

> **Note:**
>
> $Y_{11}$ is defined as the current entering the circuit at port 1 when port 2 is shorted and port 1 is excited with a 1-Volt source. $\omega$ is the frequency in radians/second. *Freq* is an internal variable, specifically the sweep frequency in Hz (or cycles/second). The $2\pi$ factor converts the frequency to radians/second.

The *Report* dialog box should still be open from the preceding topic. If not, repeat the first step of the previous procedure to reopen it.

1. In the *Report* dialog box, click **Output Variables**.
2. Specify the following settings to define *L* in the *Output Variables* dialog box that appears:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Type **L** in the **Name** text box.
b. Select **Terminal Y Parameter** from the **Category** drop-down menu.
c. In the **Quantity** list, ensure that **Yt(Source1_T1, Source1_T1)** is selected.
d. In the **Function** list, select **im**.
e. At the bottom of the *Quantities* section, click **Insert Into Expression**.
f. Click inside the **Expression** text box to place the cursor between the first parenthesis *(* and *Yt*. Type **1/** to invert the Y-parameter:

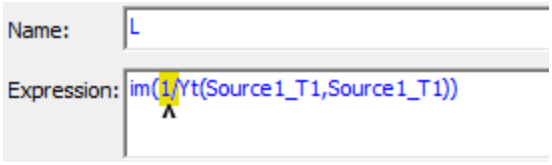



**Figure 5-17: Inverting the Y-Parameter**

g. Place the cursor at the end of the *Expression* and append it by typing **/(2*pi*freq)** to complete it.
h. Click **Add**.

The *L* expression is added to the *Output Variables* list, and it should exactly match the following figure:

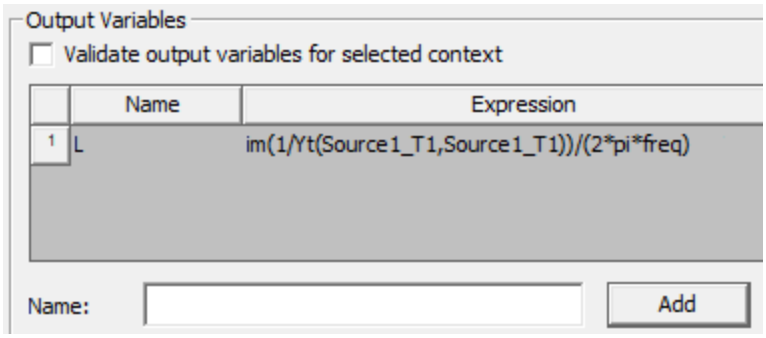



**Figure 5-18: *L* Output Variable Added**

3. Click **Done** to close the *Output Variables* dialog box.
4. In the *Report* dialog box, specify the settings shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

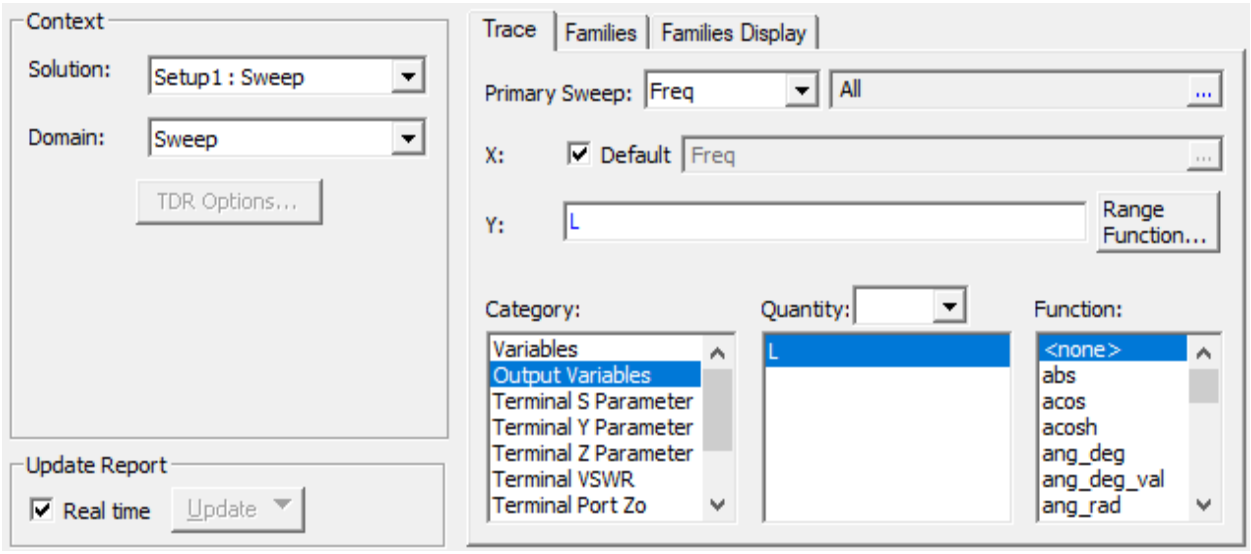



**Figure 5-19: *L* Plot Settings**

5. Click **New Report** but keep the dialog box open for now.

   The output variable plot (*L* vs. Frequency) appears in a new window:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

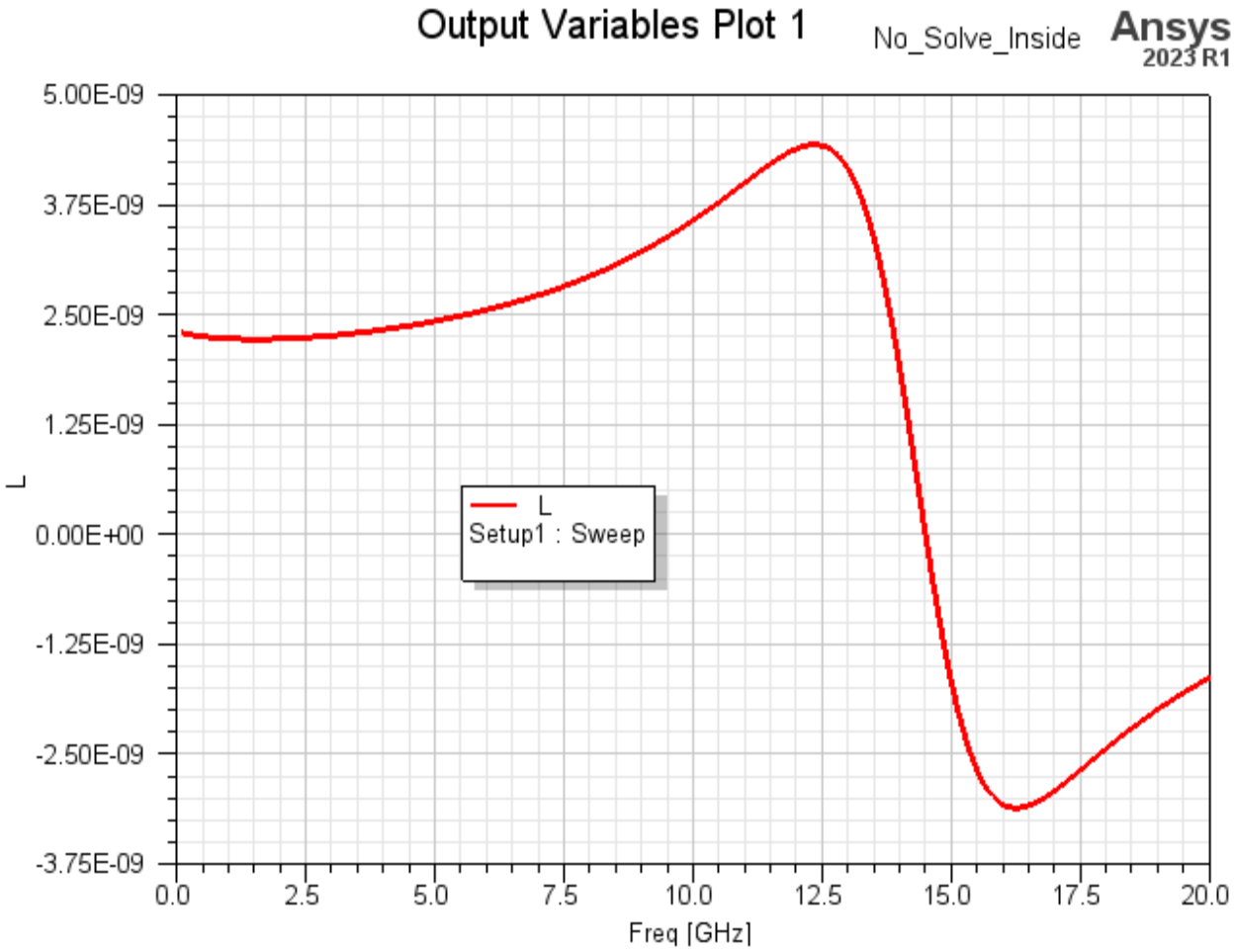



**Figure 5-20: *L* vs. Frequency Plot**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

**Observations:**

- At approximately 14.5 GHz, the inductance (*L*) goes negative. 14.5 GHz is a resonant frequency of the spiral inductor structure. At this frequency and above, the device is no longer acting as an inductor.
- The disturbance of the *L* value at and near the point of resonance is an artifact of the way that the *L* value was extracted. Remember that the assumptions used to simplify the *L* equation included the condition that operation occurs well below the resonant frequency.
- The frequency one octave below resonance is 7.25 GHz. From 0 to 7.25 GHz, the inductance is fairly stable and consistently within the range of 2.21E-9 to 2.77E-9. From 0 GHz to two octaves below resonance (3.625 GHz), the inductance remains close to 2.26E-9 (specifically, within ±0.04E-9 or ±1.8%).

## Quality Factor (*Q*) vs. Frequency:

Another metric often used to characterize spiral inductors is the quality factor (*Q*), defined as the ratio of the imaginary part of the inductor's impedance over the real part. Based on the previously stated assumptions, this factor can also be extracted from the $Y_{11}$ parameter results. The approximation is derived as follows:

$R \approx re(1/Y_{11})$

$L \approx im(1/Y_{11})/\omega$

$Q = \omega L/R \approx im(1/Y_{11}) / re(1/Y_{11})$

6. In the *Report* window, click the **Output Variables** button again.
7. Specify the following settings to define *Q* in the *Output Variables* dialog box that appears:
    a. Type **Q** in the **Name** text box.
    b. Select **Terminal Y Parameter** from the **Category** drop-down menu.
    c. In the **Quantity** list, ensure that **Yt(Source1_T1,Source1_T1)** is selected.
    d. In the **Function** list, select **im**.
    e. At the bottom of the *Quantities* section, click **Insert Into Expression**.
    f. Click inside the **Expression** text box to place the cursor between the first parenthesis *(* and *Yt*. Type **1/** to invert the Y-parameter.
    g. At the end of the *Expression*, append a forward slash (*/*):


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The expression is red because it is incomplete. The divide operator (forward slash) causes HFSS to expect another variable. Red text indicates an incorrect or incomplete expression.

h. In the **Function** list, select **re**.

i. Click **Insert Into Expression** again.

j. Click inside the **Expression** text box to place the cursor between *re(* and *Yt*. Type **1/** to invert the second Y-parameter.

k. Click **Add**.

The completed expression for *Q* appears in the *Output Variables* table:

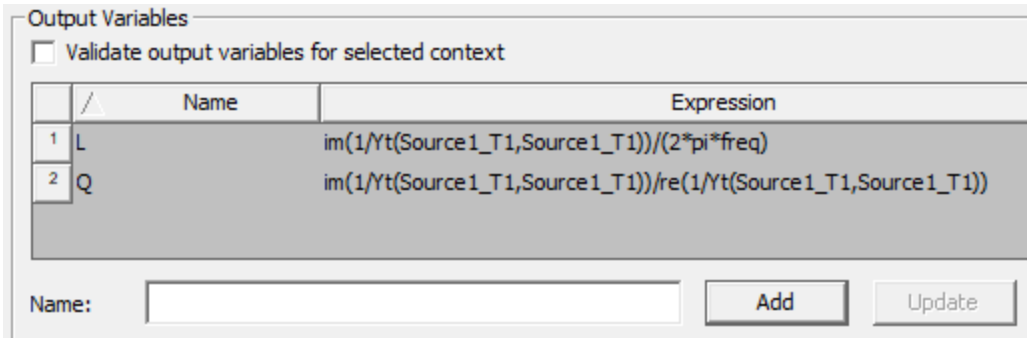



**Figure 5-21: *Q* Output Variable Added**

8. Click **Done**.

   The *Output Variables* dialog box closes.

9. Edit the settings in the *Report* dialog box as shown in the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

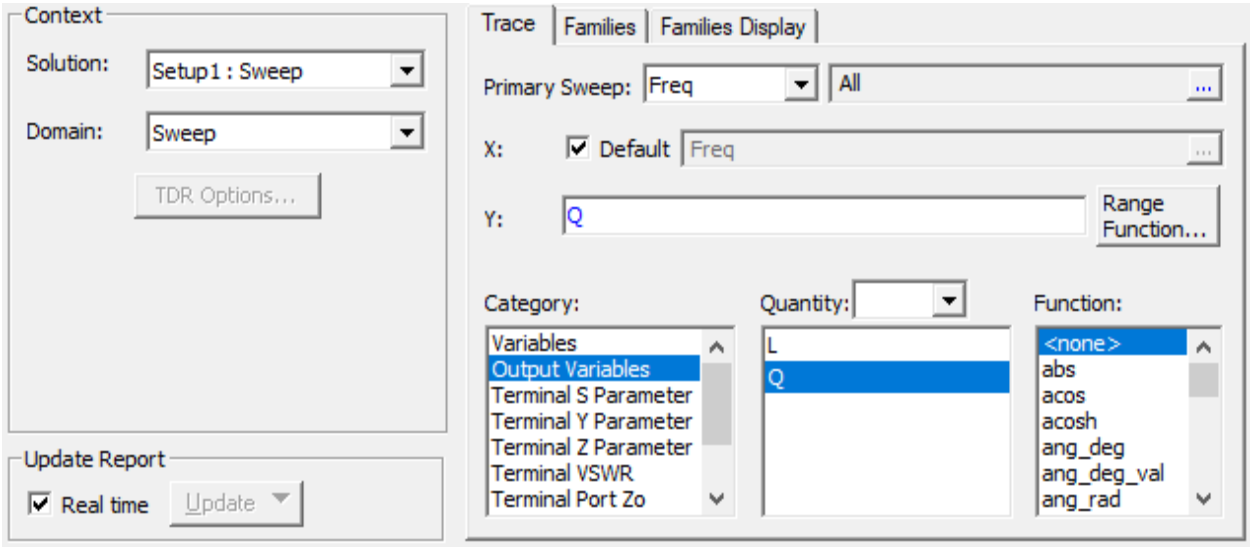



**Figure 5-22: *Q* Plot Settings**

10. Click **New Report** and then click **Close**.

    The output variable plot (*Q* vs. Frequency) appears in a new window:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

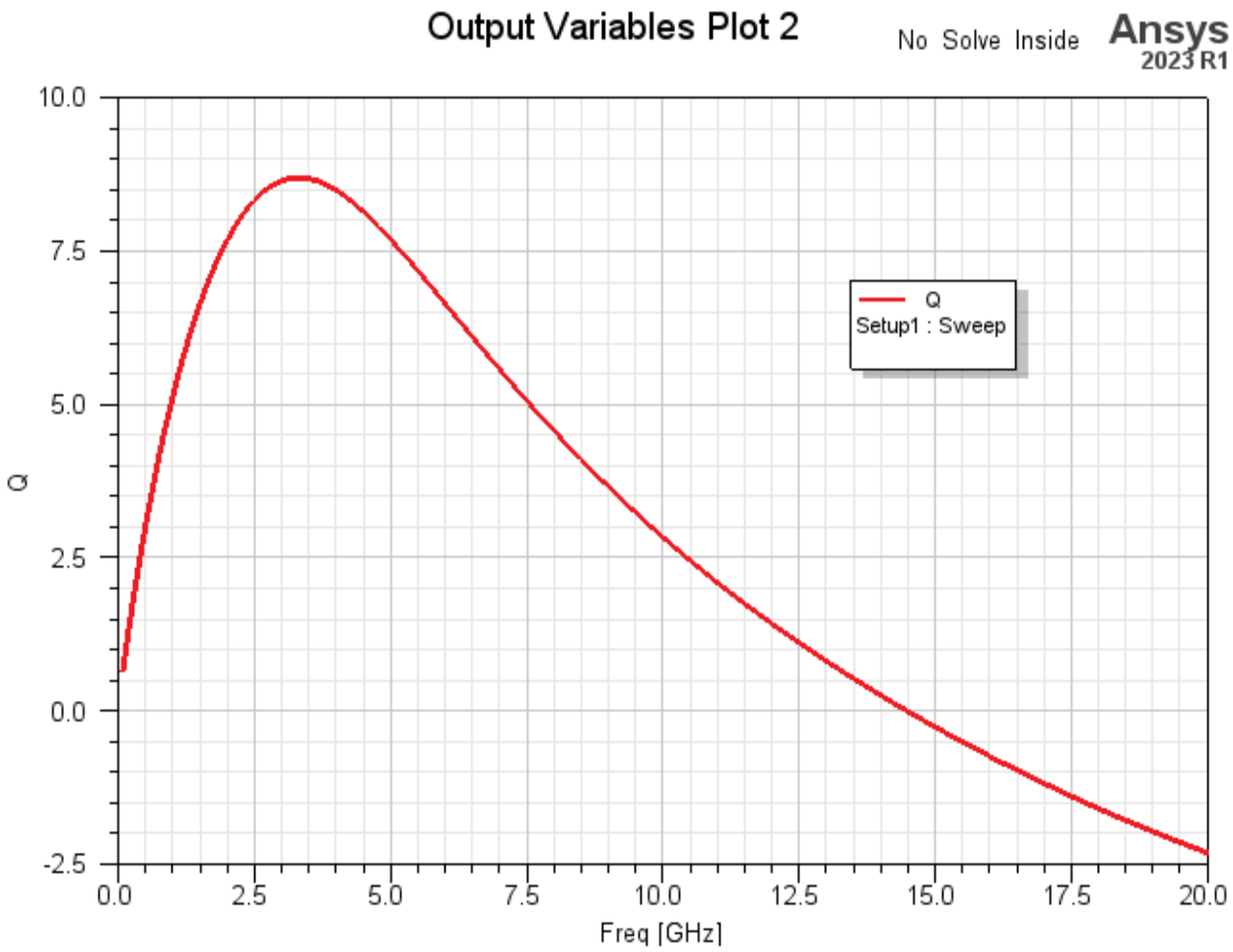



**Figure 5-23: *Q* vs. Frequency Plot**

**Observations:**

- The quality factor (*Q*) peaks at about 8.7. This peak occurs at approximately 3.3 GHz.
- *Q* becomes negative at 14.5 GHz. This is the point at which the *L* value went negative (while the resistance remained positive). Since the component is no longer acting as an inductor at and above the resonant frequency, the *Q* values in this range are not meaningful.

11. **Save** your project.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Simulate with Solve Inside Conductors

In this section, you will duplicate your HFSS design, modify the copy to enable the *Solve Inside* option for the *Spiral* conductor, and analyze the modified design. Afterward, you will compare the results with those produced without the *Solve Inside* option enabled for the spiral conductor.

By default *Solve Inside* gets automatically deselected for metals or highly conductive materials. The conductive material is represented by a boundary condition that removes the need to solve inside the metal. For most projects, we recommend that you use the default settings. When *Solve Inside* is selected, it generates tetrahedra inside the conductor, which may require a large number of elements. *Solve Inside* can be useful for low frequency analyses of electrically small projects to provide enhanced accuracy of sensitivity design parameters (such as the Q factor).

1. Right-click **No_Solve_Inside (Terminal Network)** in the Project Manager and choose **Copy** from the shortcut menu.
2. Right-click **Si_Spiral_Conductor** at the top of the Project Manager and choose **Paste**.

   A new design, *No_Solve_Inside1 (DrivenTerminal)*, appears in the Project Manager below the original one, and a new Modeler window opens.

3. Select the new design entry in the Project Manager.
4. Press **F2**, change the name to **Solve_Inside**, and press **Enter**.
5. Collapse the original design branch [ *No_Solve_Inside (TerminalNetwork)* ] and expand the new design branch [ *Solve_Inside (TerminalNetwork)* ]:

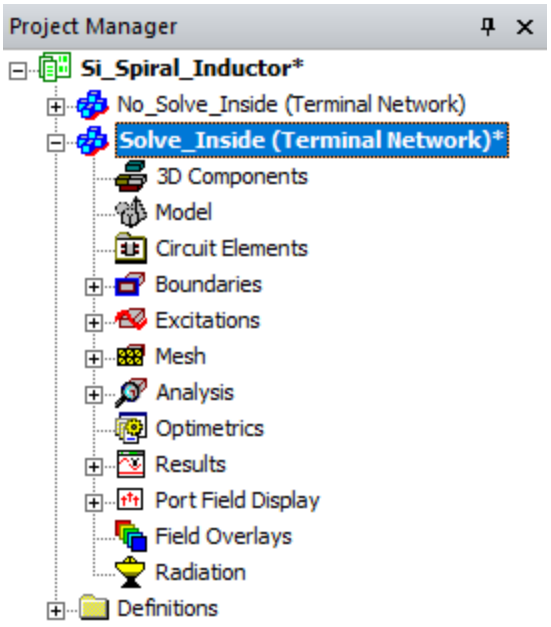



**Figure 5-24: New HFSS Design in Project Manager**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To ensure that you don't accidentally modify the wrong design, close the *Spiral_Inductor* – ***No_Solve_Inside*** – *Modeler* window. You can use the **Window** menu to select it and then click the **X** button:
   - To close a maximized window, click this button near the upper right corner of the program screen:

**Figure 5-25: Closing a Maximized Window**

   - To close a floating (non-maximized) window, click this button in the upper right corner of the window:

7. Use the **Window** menu to make the *Spiral_Inductor* – ***Solve_Inside*** – *Modeler* window active (bringing it to the foreground).
8. Under *Model* > *Solids* > *My Metal* in the History Tree, select **Spiral**.

   The object's attributes are displayed in the docked Properties window.
9. In the docked Properties window, select the **Solve Inside** option.

> **Note:**
>
> The *Message Manager* window displays the following message: *Solving inside a solid with high conductivity may require a large mesh*.

10. In the new design branch of the Project Manager, right-click **Analysis** and choose **Analyze All** from the short-cut menu.

    This design variant will take four or five times longer to solve than the original design, due to the adaptive meshing of the spiral conductor and the higher element count.

# Results with Solve Inside

The new HFSS design has all of the output variables and plots that were defined for the original design. However, the plot windows that are already open remain associated with the design in which they were generated. To display the plots for the new design, you simply have to double-click plot headings under *Results* in the Project Manager.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before displaying the plots for the modified design, let's look at the Convergence plot and Mesh Statistics.

1. Under *Solve_Inside (Terminal Network)* > *Analysis* in the Project Manager, right-click **Setup1** and choose **Convergence**.
2. For the **View** option, choose **Plot**.

   You can see that the solution converged in nine passes (one fewer that the previous analysis), and with a similar Max Mag. Delta S achieved.

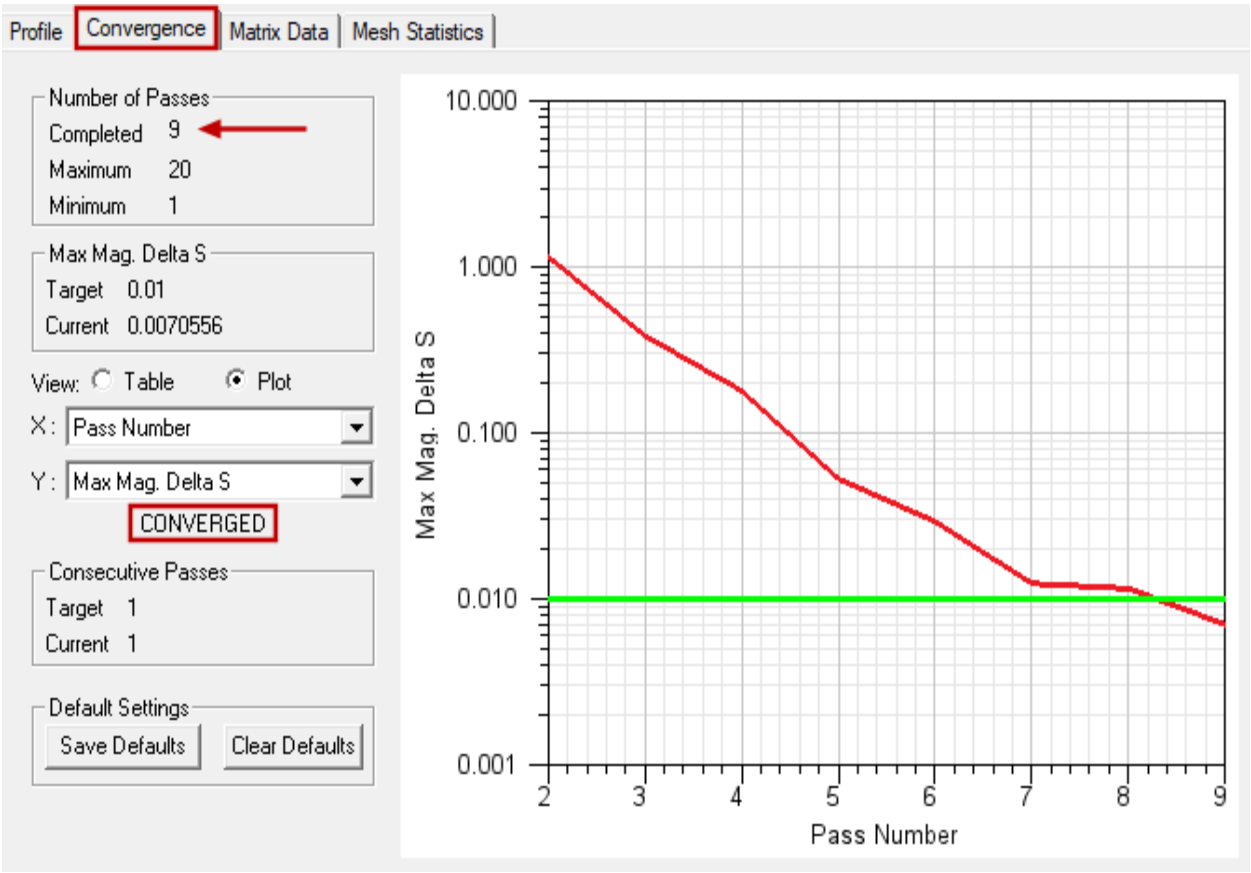
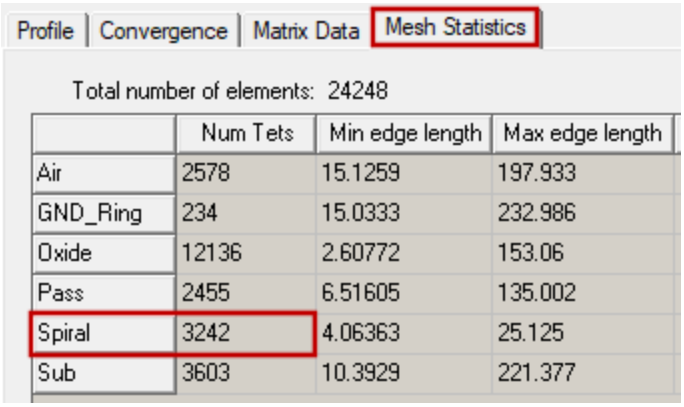



**Figure 5-26: Convergence Plot – Solve Inside Enabled for Spiral**

3. Select the **Mesh Statistics** tab.

   The element count for the *Spiral* object is approximately 2.5 times that of the previous analysis (3242 versus 1297 for the previous analysis).


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.




| | Num Tets | Min edge length | Max edge length |
|---|---|---|---|
| Air | 2578 | 15.1259 | 197.933 |
| GND_Ring | 234 | 15.0333 | 232.986 |
| Oxide | 12136 | 2.60772 | 153.06 |
| Pass | 2455 | 6.51605 | 135.002 |
| Spiral | 3242 | 4.06363 | 25.125 |
| Sub | 3603 | 10.3929 | 221.377 |

**Figure 5-27: Mesh Statistics – Solve Inside Enabled for Spiral**

> **Note:**
>
> Your results may be different since meshing and solution convergence behavior can vary somewhat between different computing platforms and different software versions.

4. Click **Close** to dismiss the *Solutions* dialog box.
5. Under *Solve_Inside (Terminal Network)* > *Results* in the Project Manager, double-click each of the following plots to generate them for the *Solve_Inside* design:
   a. **Terminal S Parameter Plot 1**
   b. **Output Variables Plot 1**
   c. **Output Variables Plot 2**

   The following plots appear in new windows:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

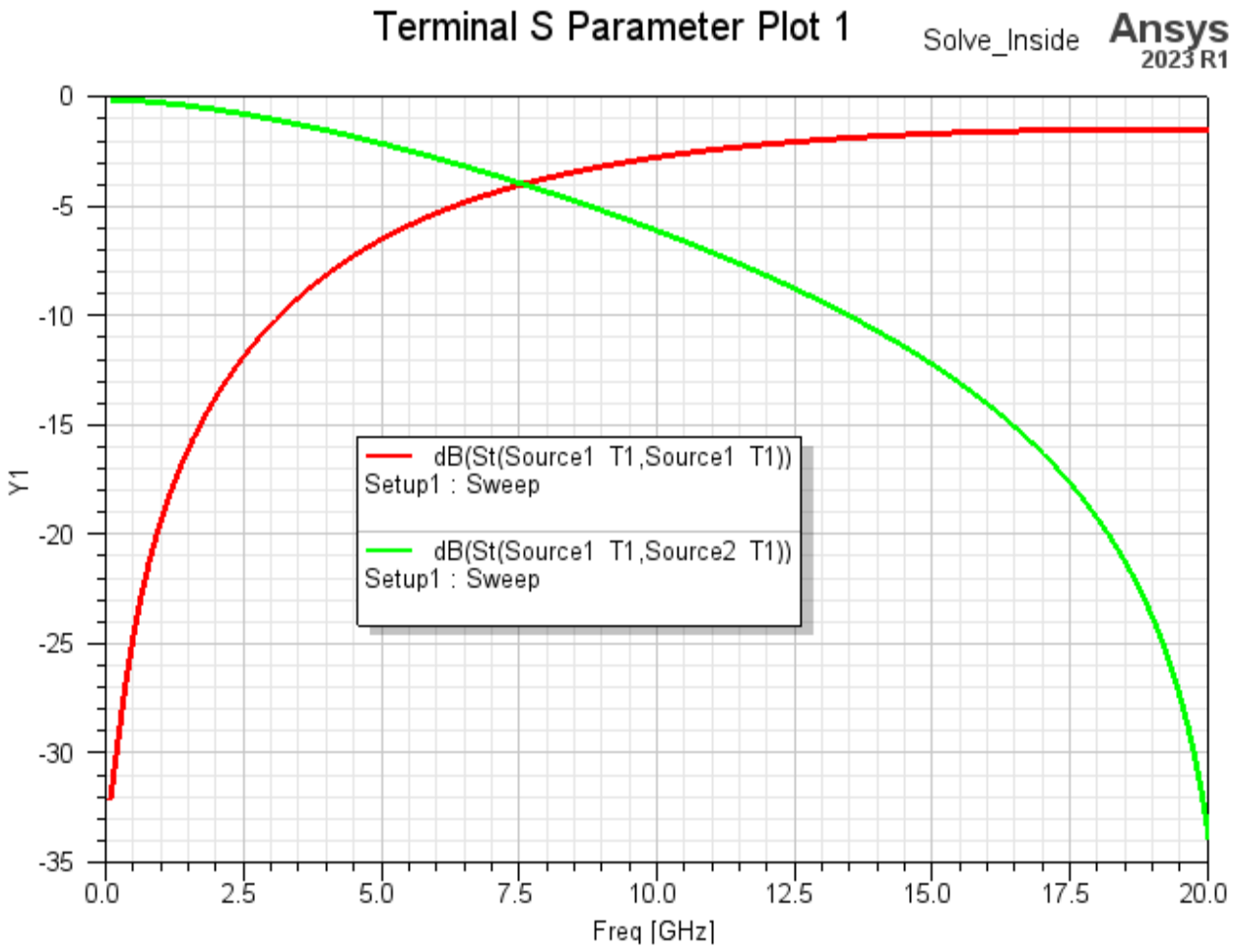



**Figure 5-28: S-Parameters vs. Frequency Plot (*Solve_Inside*)**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

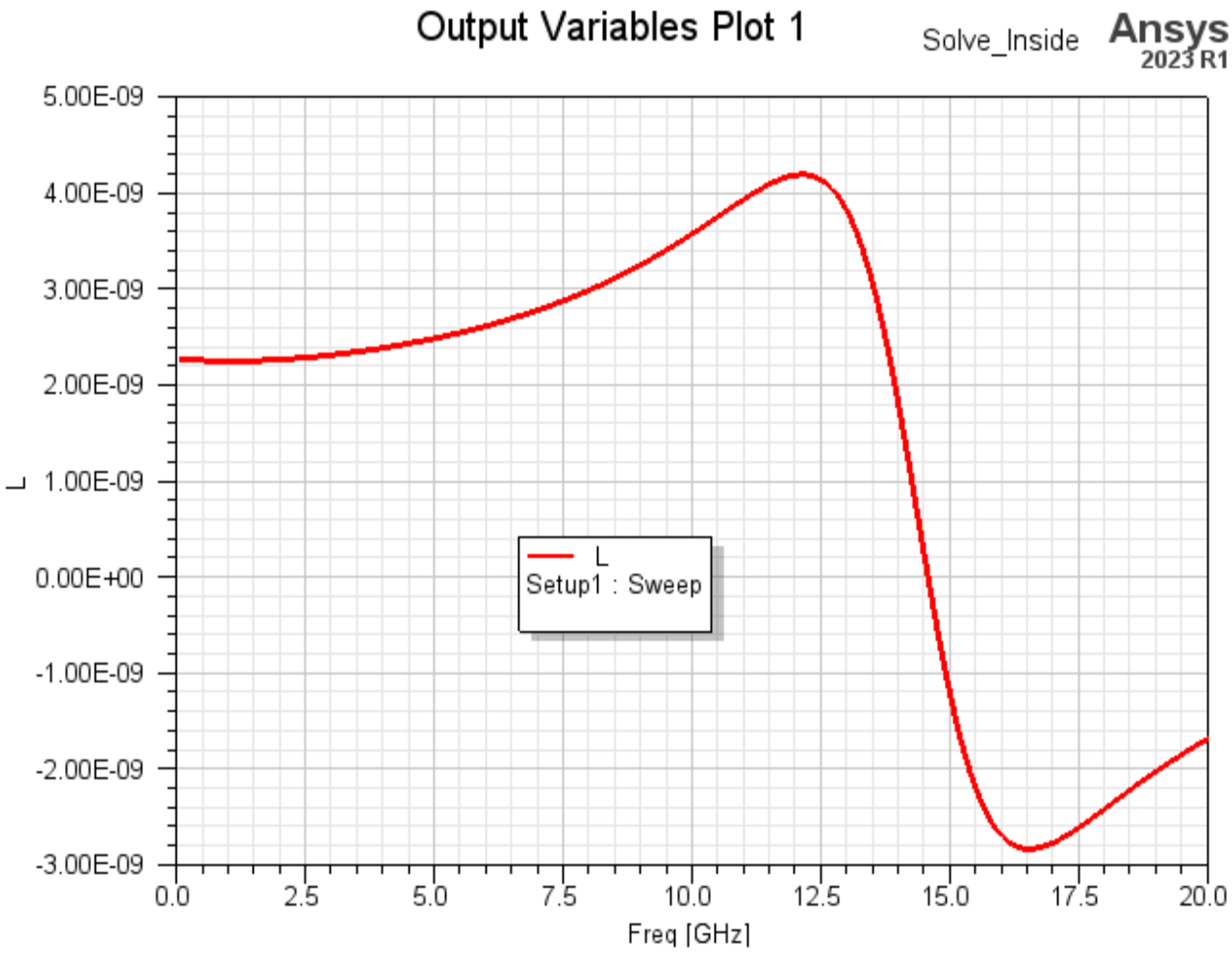



**Figure 5-29: *L* vs. Frequency Plot (*Solve_Inside*)**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

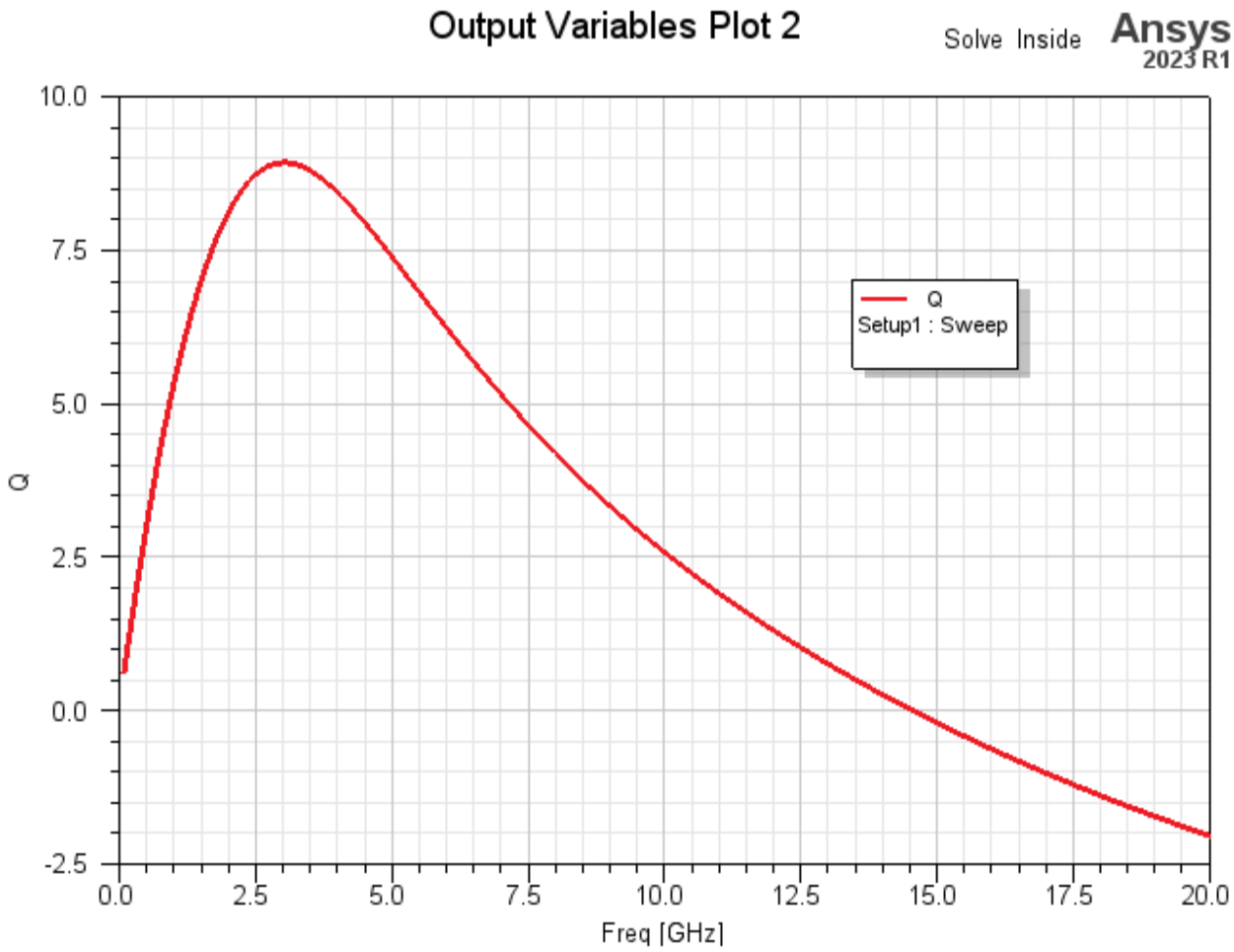



**Figure 5-30: *Q* vs. Frequency Plot (*Solve_Inside*)**

**Observations:**

- Qualitatively, all three plots from the *Solve_Inside* design are similar to the corresponding plots from the *No_Solve_Inside* design.
- The quantitative difference are somewhat difficult to perceive because the Y scales differ between the plots from each design. The scale settings could be adjusted to be the same for both designs. You could also add markers to indicate the numerical value of the results at designated points. However, there's an easier way to compare the results of two or more plots, which will be demonstrated in the next topic.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# Direct Comparison of Results

For the best comparison of results from two different plots, the traces should be viewed within the same plot and using the same X and Y scales. Ansys Electronics Desktop allows you to copy plot data from traces in one plot and paste them into another plot for direct comparison using identical scaling parameters. You will create three new plots in the second design, rename them, and copy and paste in the traces from the first design for comparison.

1. Under *Solve_Inside (DrivenTerminal)* > *Results* in the Project Manager, select **Terminal S Parameter Plot 1, Output Variable Plot 1,** and **Output Variable Plot 2** (holding down Ctrl to select multiple items).
2. Right-click on one of the selected plot titles and choose **Copy Definition** from the shortcut menu.
3. Right-click **Results** and choose **Paste**.

   Three additional plots appear in the Results branch with *"_1"* appended to each title.
4. Right-click **Terminal S Parameter Plot 1_1**, choose **Rename**, change the name to **S Parameter Results Comparison**, and press **Enter**.
5. Under *No_Solve_Inside* > *Results* > *Terminal S Parameter Plot 1* in the Project Manager, select **both traces** [*dB(St(Source1_T1,Source1_T1))* and *dB(St(Source1_T1,Source2_T1))*.
6. Right-click on one of the two selected traces and choose **Copy Data**.
7. Under *Solve_Inside* > *Results* in the Project Manager, right-click **S Parameter Results Comparison** and choose **Paste**.

   Two traces are added to the *S Parameter Results Comparison* plot. The red and green traces represent the *Solve_Inside* results, and the blue and orange traces represent the *No_Solve_Inside* results.

   > **Note:**
   >
   > If the color of any trace does not match the colors in this description, select the trace in the legend and change the **Color** selection in the docked *Properties* window.
   >
   > Where a trace is directly over another, you will only see the color of the most recently added one. If you point to a row in the legend, the associated trace will turn green to help you identify overlapping curves.
8. Right-click in the new plot window and choose **Add Note**. Then, in the *Add Note* dialog box, type the note exactly as shown in the following figure and click **OK** (stretch the window width as needed):


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

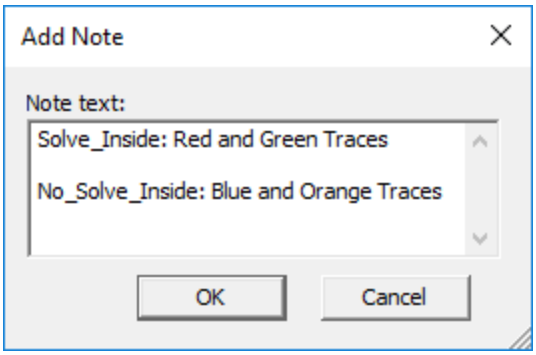



**Figure 5-31: Comparison Plot Notation**

9. Click and drag the borders of the yellow notation to resize it. Also, position the legend and notation to produce a comparison plot similar to the following figure:

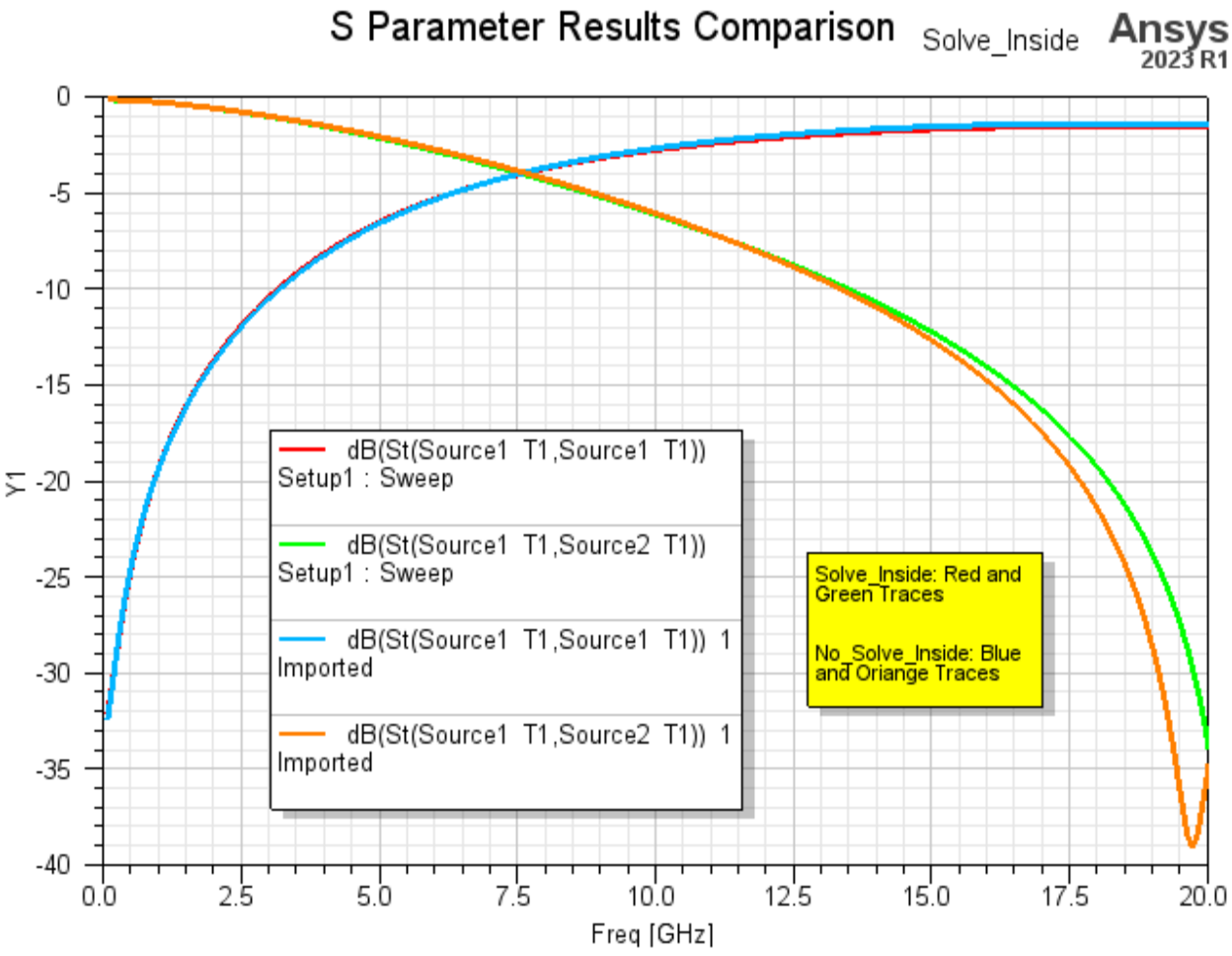



**Figure 5-32: S Parameter Results Comparison Plot**


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now you can clearly see the divergence above 13 GHz of the *dB(St(Source1_T1,Source2_T1))* results between the two designs.

10. In the same manner as detailed in steps 4 through 9, rename **Output Variables Plot 1_1** as **L Results Comparison**, copy and paste the corresponding trace from the first design, and add and position a suitable notation.

    The resultant plot should look like the following figure:

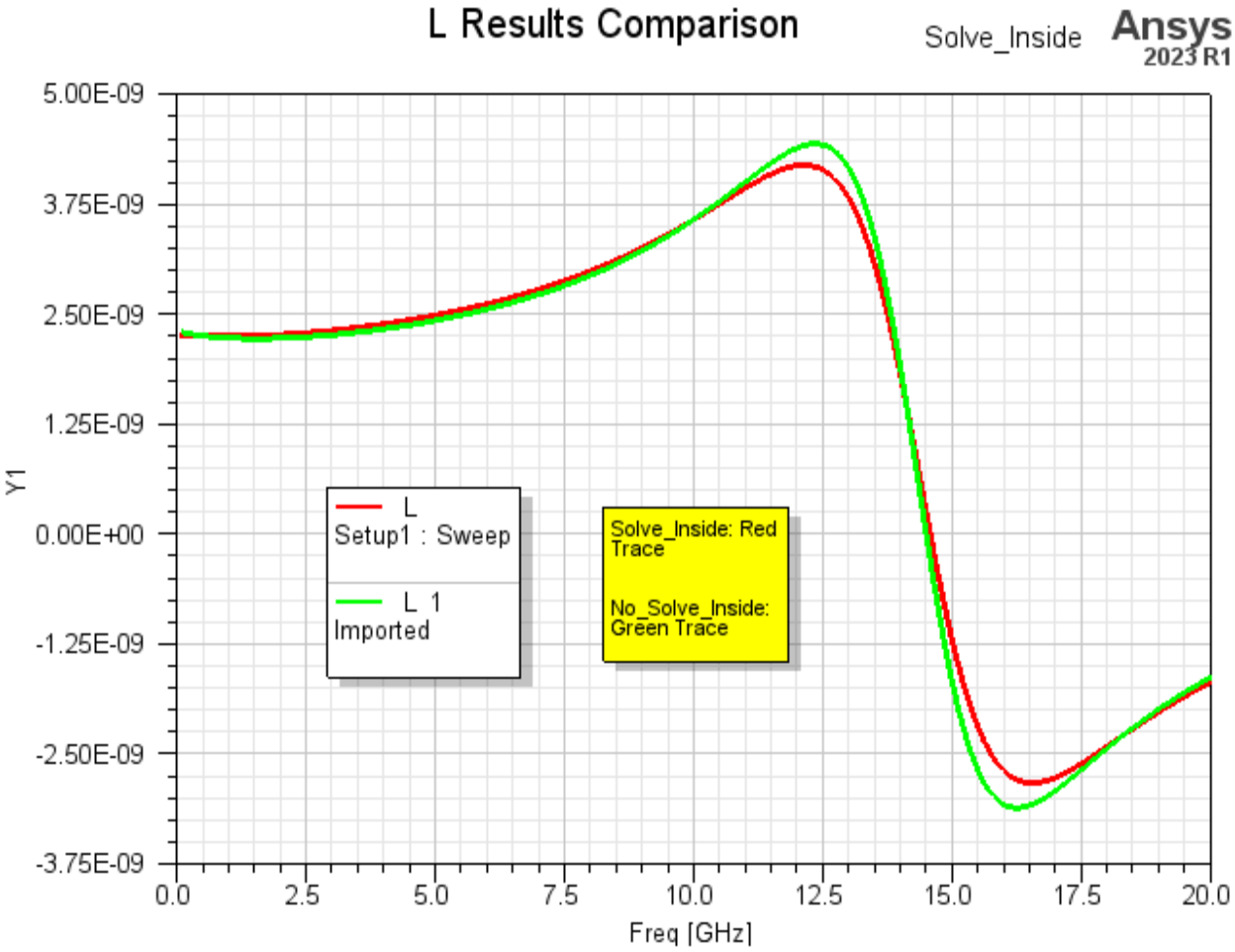



**Figure 5-33: *L* Results Comparison Plot**

11. Again, in the same manner as detailed in steps 4 through 9, rename **Output Variables Plot 2_1** as **Q Results Comparison**, copy and paste in the corresponding trace from the first design, and add and position a suitable notation:

    The resultant plot should look like the following figure:


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

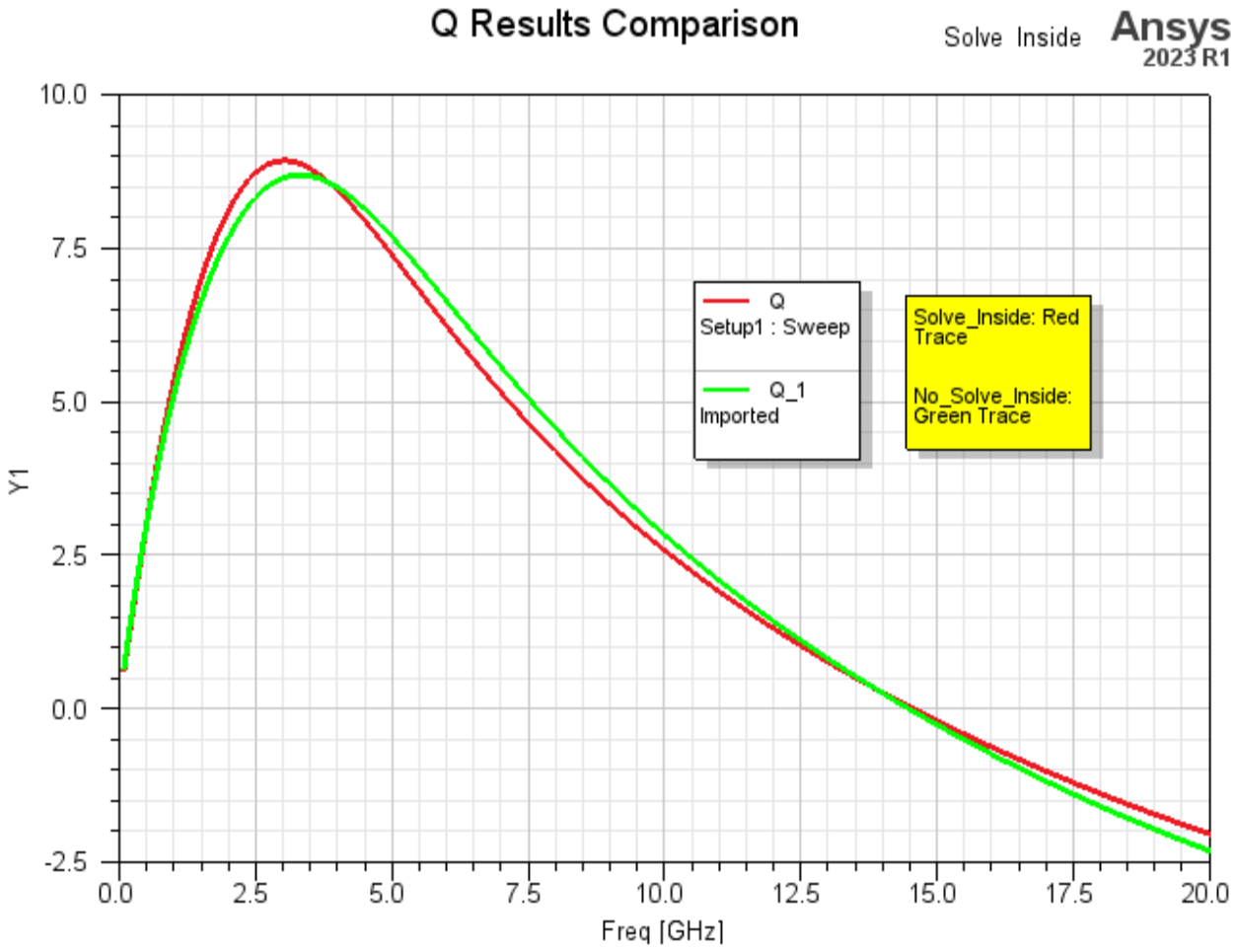



**Figure 5-34: *Q* Results Comparison Plot**

12. **Save** your project.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

# 6 - Optionally, Restore Current View Orientations

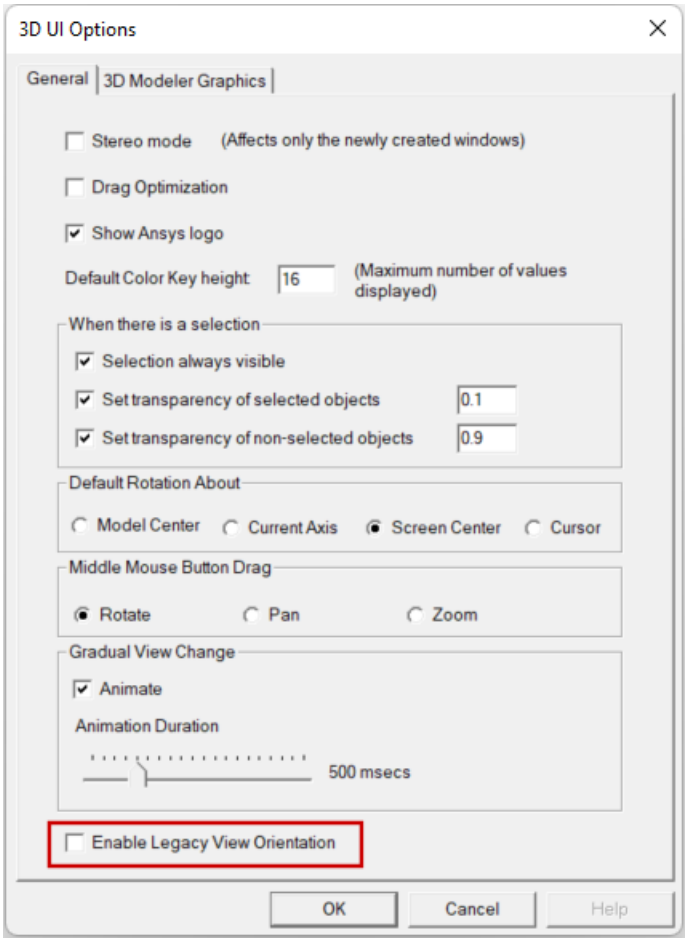
You have completed this getting started guide.

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.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can now save and close this project.


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.