Getting Started with HFSS: Silicon Spiral Inductor

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 18.0
December 2016

Conventions Used in this Guide

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

- 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 the word copy must be **typed**, then a space must be typed, and then **file1** must be typed.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. 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** *file name*” the word **copy must be typed, then** a space must be typed, and then name of the file must be typed.
- 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 the F1 key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,

  “On the Draw menu, click Line  ” means that you can click the Draw Line toolbar button to execute the Line command.

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 online help from the menu bar, click **Help** and select from the menu:

- **HFSS Contents -** click here to open the contents of the online help.
- **HFSS Search -** click here to open the search function of the online help.

Context-Sensitive Help

To access online help from the user interface, do one of the following:

- To open a help topic about a specific menu command, press **Shift+F1**, and then click the command or toolbar icon.
- To open a help topic about a specific dialog box, open the dialog box, and then press **F1**.

Table of Contents

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, simulate, and analyze a 2.5 turn spiral inductor.

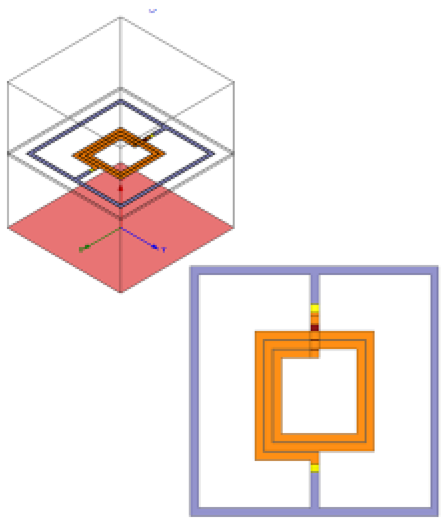


Figure 1-1 Spiral Inductor

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

- **Spiral:** 2.5T, W=15um, S=1.5um, Rad=60um.
  M6, 2um, σ= 2.8e7 S/m.
- **Underpass:** M5, 0.5um, σ= 2.8e7 S/m.
- **Stackup:** Passivation**:** 0.7um, εr = 7.9.
- **Oxide:** 9.8um, εr = 4.0.
- **Substrate:** 300um.
  εr = 11.9, σ= 10 S/m.

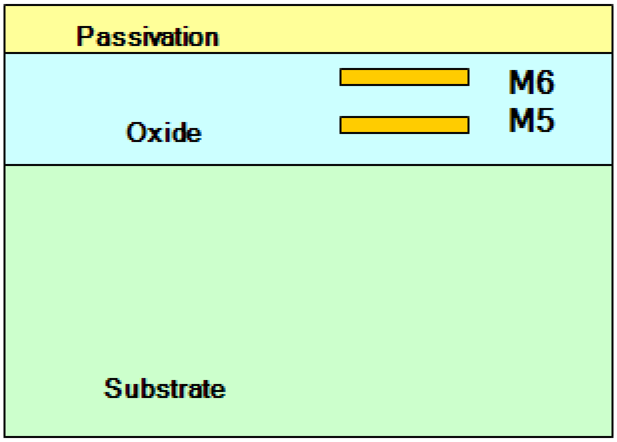


Figure 1-2 Passivation, Oxide and Substrate

2 - Set Up the Project

This chapter contains the following topics:

- Launch HFSS
- Set Tool Options
- Insert HFSS design
- Set Model Units(cm)
- Set Solution Type(Terminal)

Launch Electronics Desktop

Store a shortcut of the ANSYS Electronics Desktop application on your desktop.

1. Double-click the **ANSYS Electronics Desktop**icon to launch the application.

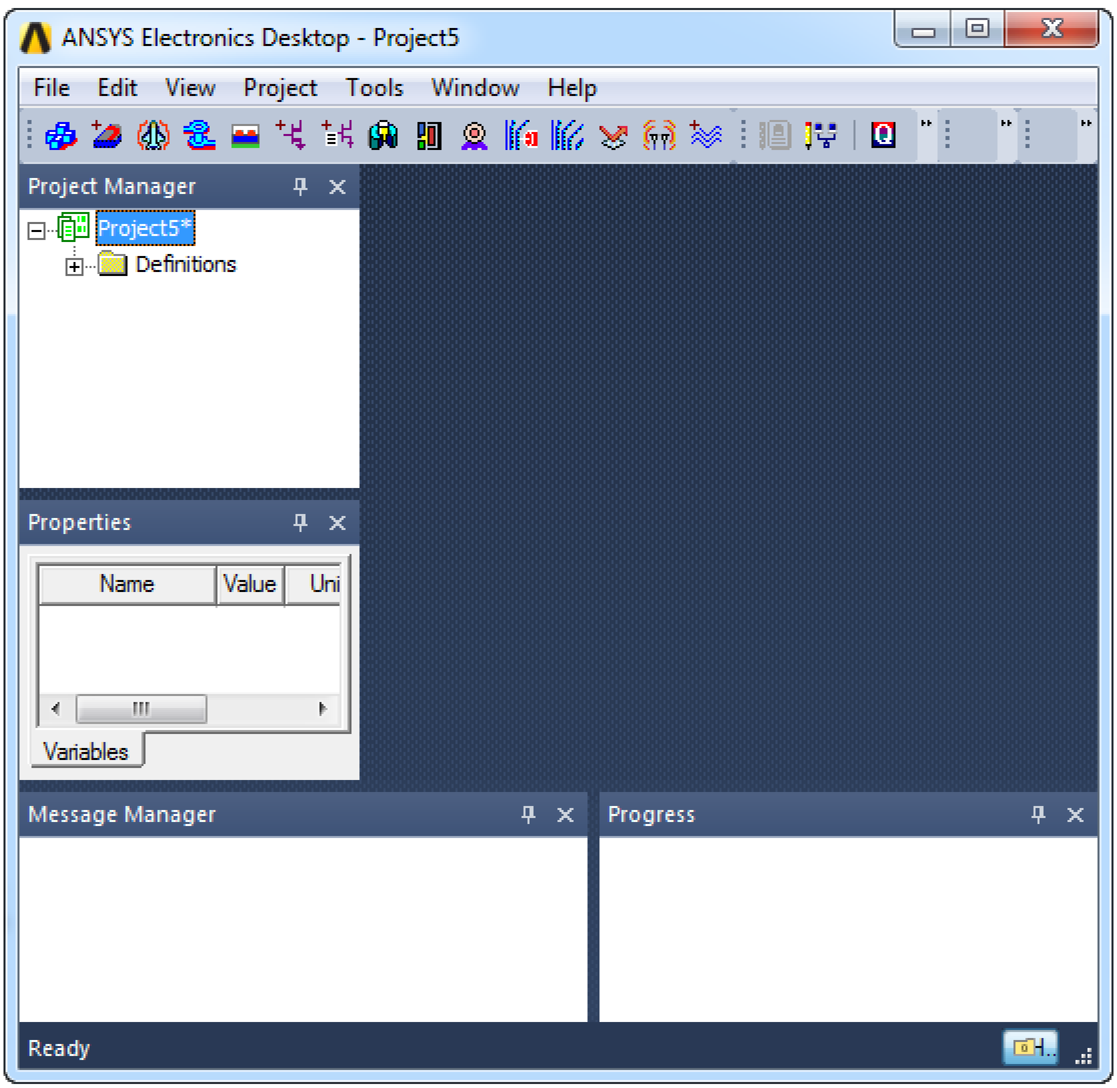


Figure 2-1 Electronics Desktop launched

Note If the application does not list the folder, go to **File** and click **New**. If the **Project Manager** window does not appear, go to **View** and enable it.

Set Tool Options

Verify the options under the **Tools** menu as follows:

1. Go to **Tools>Options>General Options.**

   The **Options** dialog box appears.

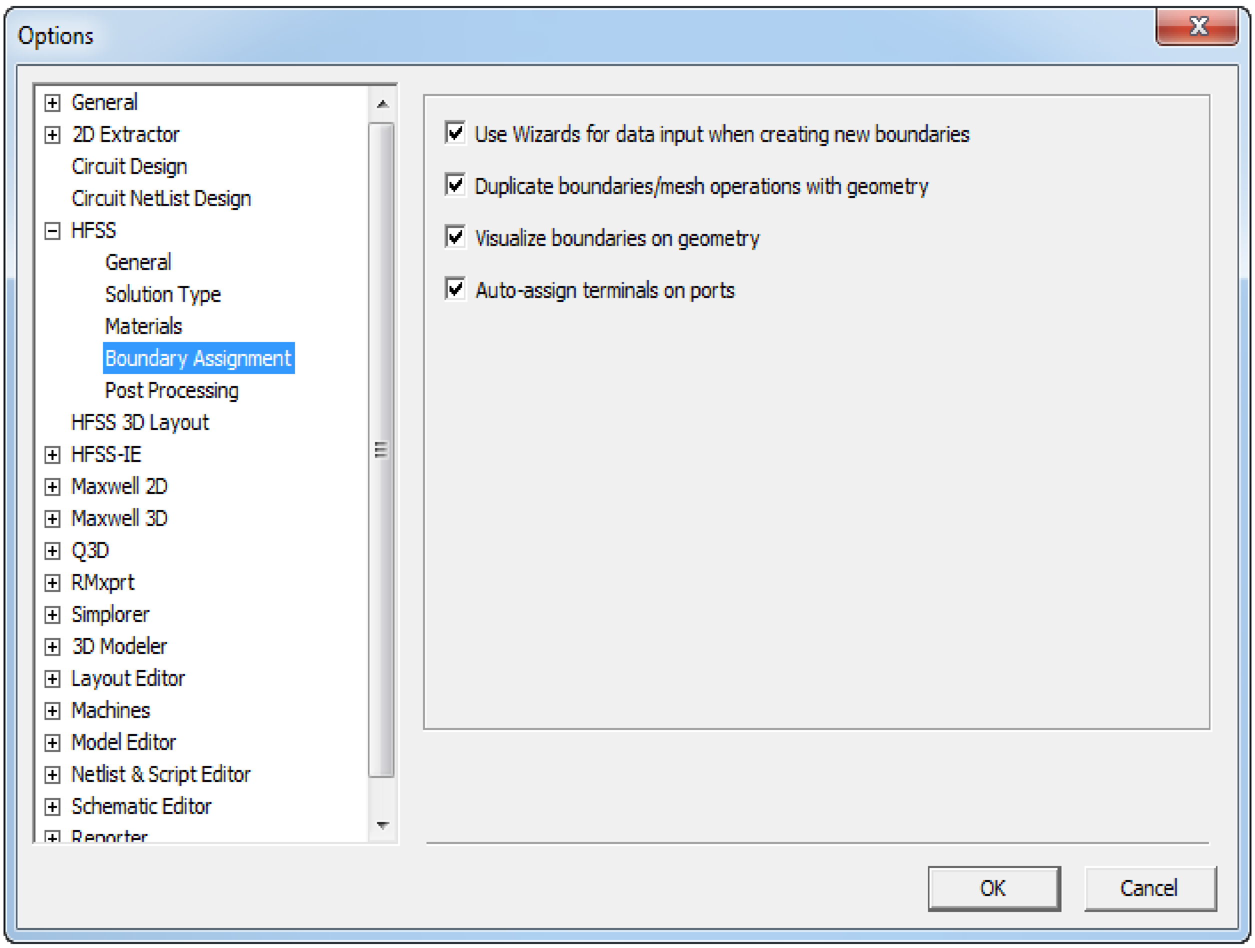


Figure 2-2 Assignment Options

2. Click **Boundary Assignment** to ensure all **Assignment Options** are checked.
3. Click **3D Modeler Options**.

   The **Modeler Options** dialog box appears.
4. Click **Drawing** and ensure the **Automatically cover closed polylines** option is selected.

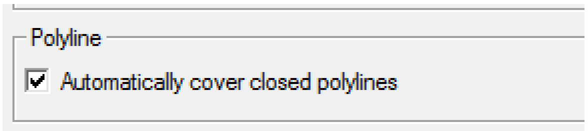


Figure 2-3 The option on the Operation tab

5. Again, on the **Drawing** tab check **Edit properties of new primitives** and click **OK.**

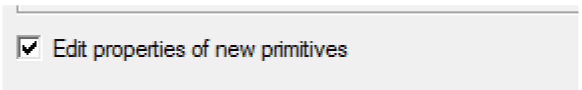
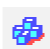


Figure 2-4 option on the Drawing tab

Note This option causes a **Properties** dialog box to appear automatically whenever you create a new object.

Insert HFSS design

The  icon represents the **Insert HFSS design** (**IHd**) option.

1. Expand the project tree.
2. If **IHd** is present, proceed to rename and save the project and if it is absent click the **IHd** icon to include it.

Note Inclusion of **IHd** modifies the project and hence the asterisk appears on **Project*n***.

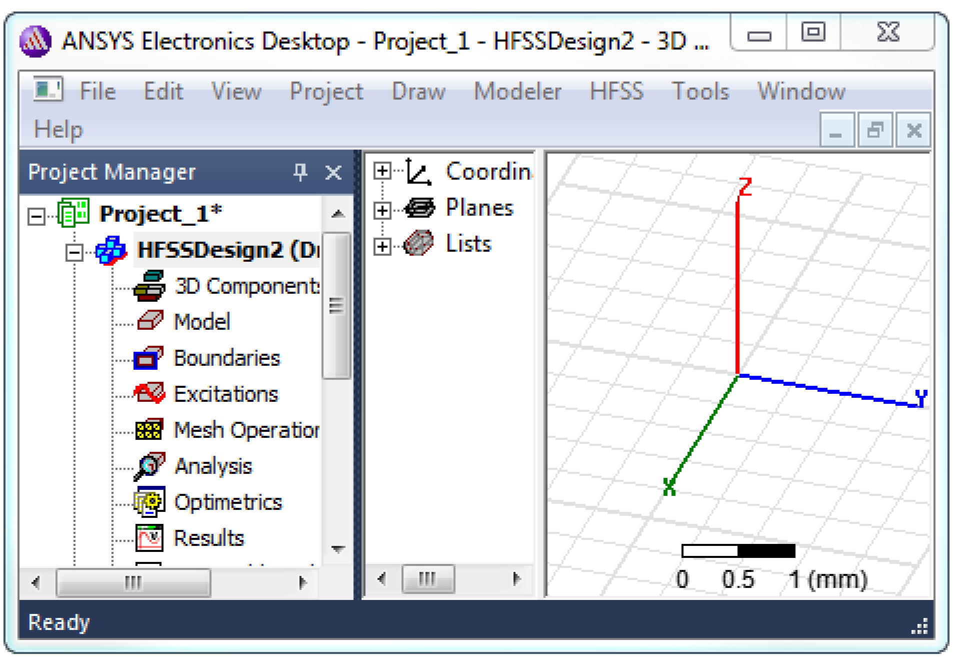


Figure 2-5 IHd included

3. Click **Project_n***, hit **F2**, rename the project as *si_spiral_inductor*, and save it.

Set Model Units

Set the units for the geometric model as follows:

1. On the HFSS toolbar, click **Modeler> Units**.
   The **Set Model Units** dialog box appears.
2. Select the unit as **um** and click **OK.**

Figure 2-6 Set Model Units dialog

Set Solution Type

To set the solution type:

1. On the toolbar, click **HFSS> Solution Type**
   The **Solution Type** dialog box appears.

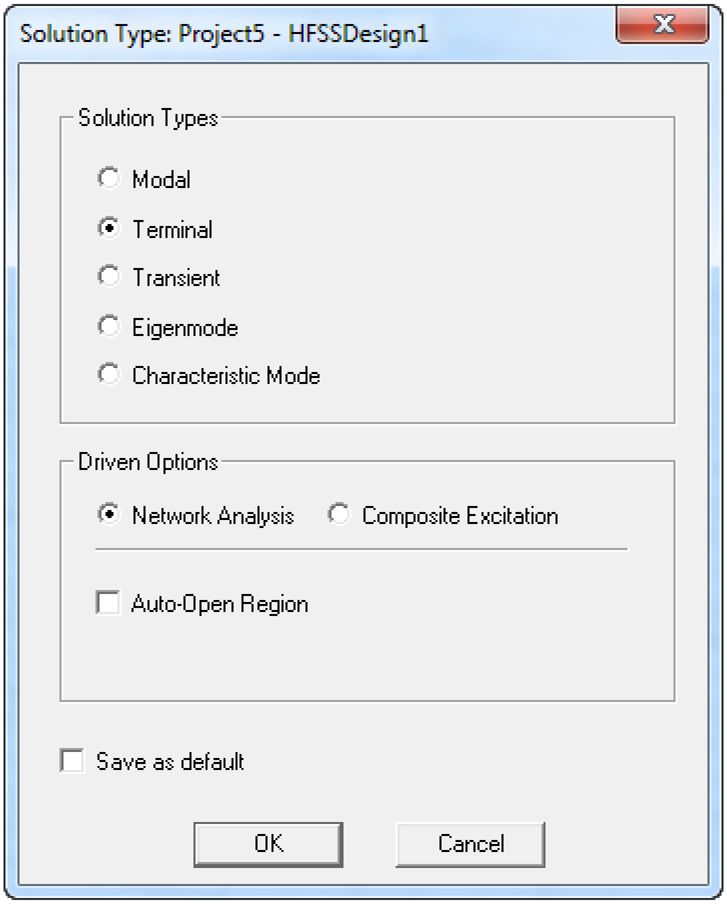


Figure 2-7 Solution Type dialog

2. Select **Driven Terminal** and click **OK.**

   Note Driven Terminal calculates 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 - Set Up Si Spiral Inductor

This chapter describes how to build the 3D spiral inductor model in HFSS.

This chapter contains the following topics:

- Create 3D Model for Dielectrics
- Create Substrate
- Create Oxide
- Create Passivation
- Create Air Body
- Assign Radiation Boundary
- Create Ground
- Assign Perfect E Boundary to the Ground
- Create Spiral Inductor Geometry
- Assign Thickness to the Spiral
- Create Underpass
- Create Via1
- Create Via2
- Create Feed
- Unite Spiral Objects
- Solve Inside Conductors
- Seed Mesh Conductors Set for Solve Inside
- Create Ground Ring
- Create Inner Ring
- Complete The Ring
- Create Extension1
- Create Extension2
- Create Source1
- Create Source2
- Group the Conductors
- Assign Excitation for Source1
- Assign Excitation for Source2

Create 3D Model for Dielectrics

To create the 3D model, you must draw a number of objects. The following sections contain the steps to create the geometry.

Create Substrate

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

1. **Click Draw>Box.**

   The cursor is accompanied by a black square box.
2. Click inside the Modeler window to establish the x,y axes and drag the mouse to draw a rect-angle
3. Click the mouse to establish the z axis and drag the mouse along the z-axis to draw the height.
4. Click the mouse again, to complete the box.

   The **Properties** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	Global		
Position	-270 ,-270 ,0	um	-270um , -270um , ...
XSize	540	um	540um
YSize	540	um	540um
ZSize	300	um	300um

Figure 3-1 Properties dialog

5. Edit the fields in the **Command** dialog box as in "View/Edit Material dialog " on the facing page.
6. On the **Attribute**, enter *Sub* in the **Name** field and from the **Materials** drop-down menu, select **Edit.**

   The **Select Definition** dialog box appears.
7. Click **Add Material** and edit the fields in the dialog box as shown in the following figure.

   Note: Mainly change relative permittivity to 11.9 and bulk conductivity to 10 siemens/m.

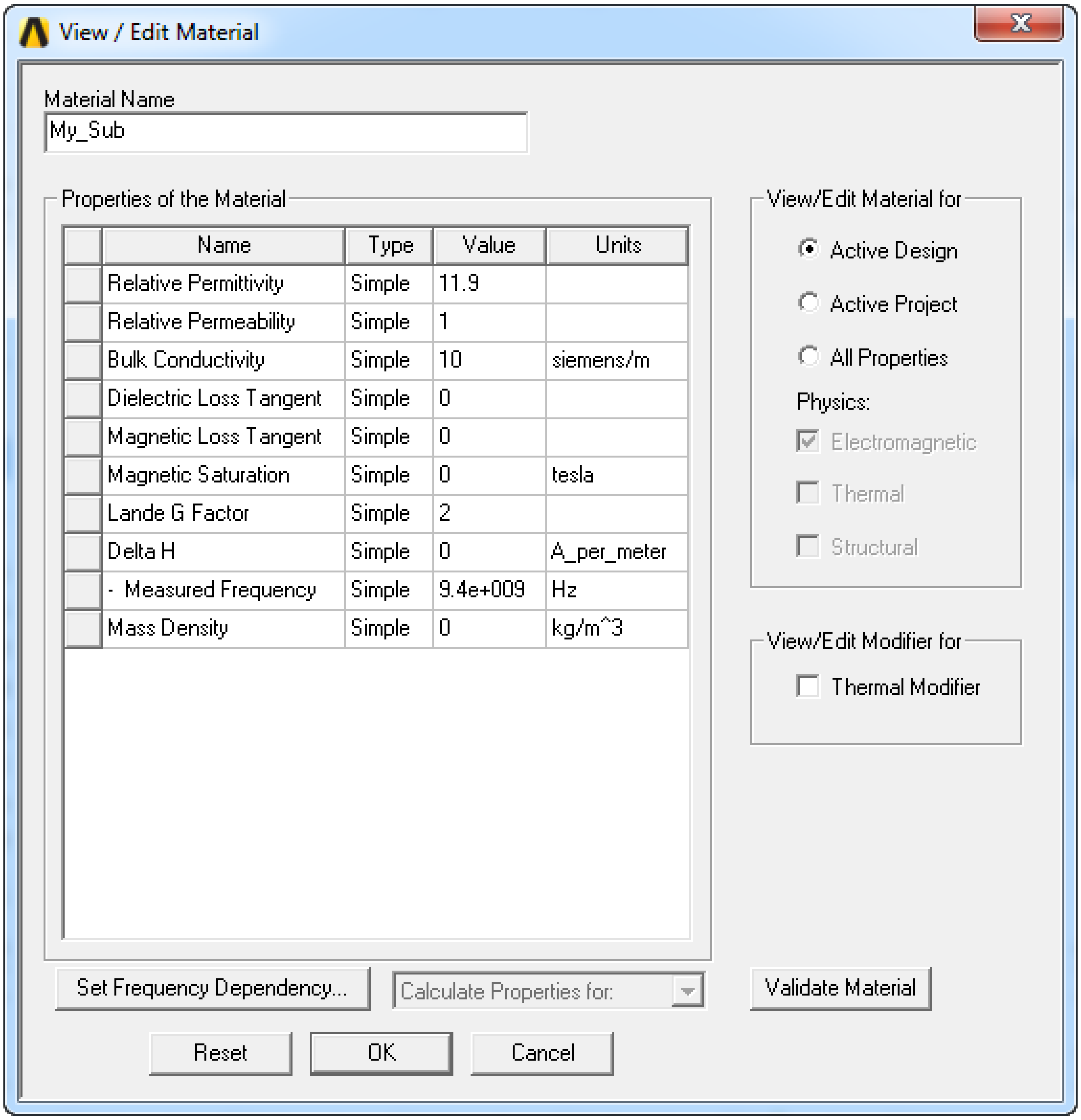


Figure 3-2 View/Edit Material dialog

8. Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog boxes to exit.
9. Do **Ctrl+D** to fit the view.

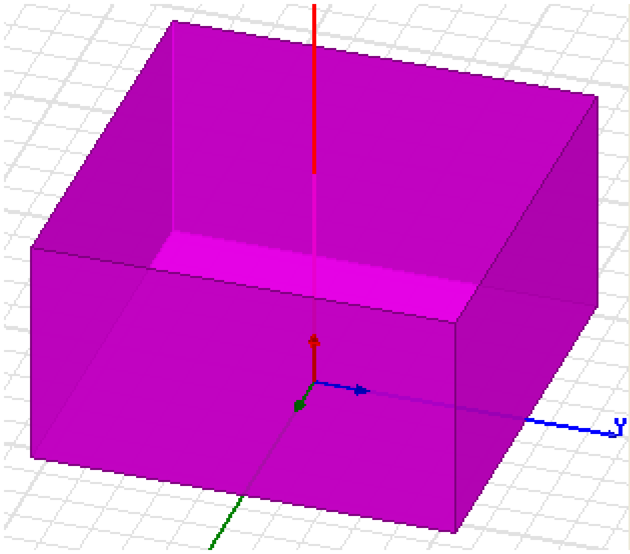


Figure 3-3 Substrate created

Create Oxide

To create the oxide, first draw a box and specify its size and location as follows:

1. Draw the box freehand.

   The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate System	Global		
	Position	-270 ,-270 ,300	um	-270um , -270um , 300um
	XSize	540	um	540um
	YSize	540	um	540um
	ZSize	9.8	um	9.8um

Figure 3-4 Command dialog for Oxide

2. On the **Command** tab, edit the fields as in "Command dialog for Oxide" above and click **Attribute** and rename box to: *Oxide*
3. From the **Materials** drop-down menu, select **Edit.**

   The **Select Definition** dialog box appears.
4. Click **Add Material** and edit the fields as in "View/Edit Material dialog" on the facing page.

   Note The **View/Edit Material** dialog box appears.

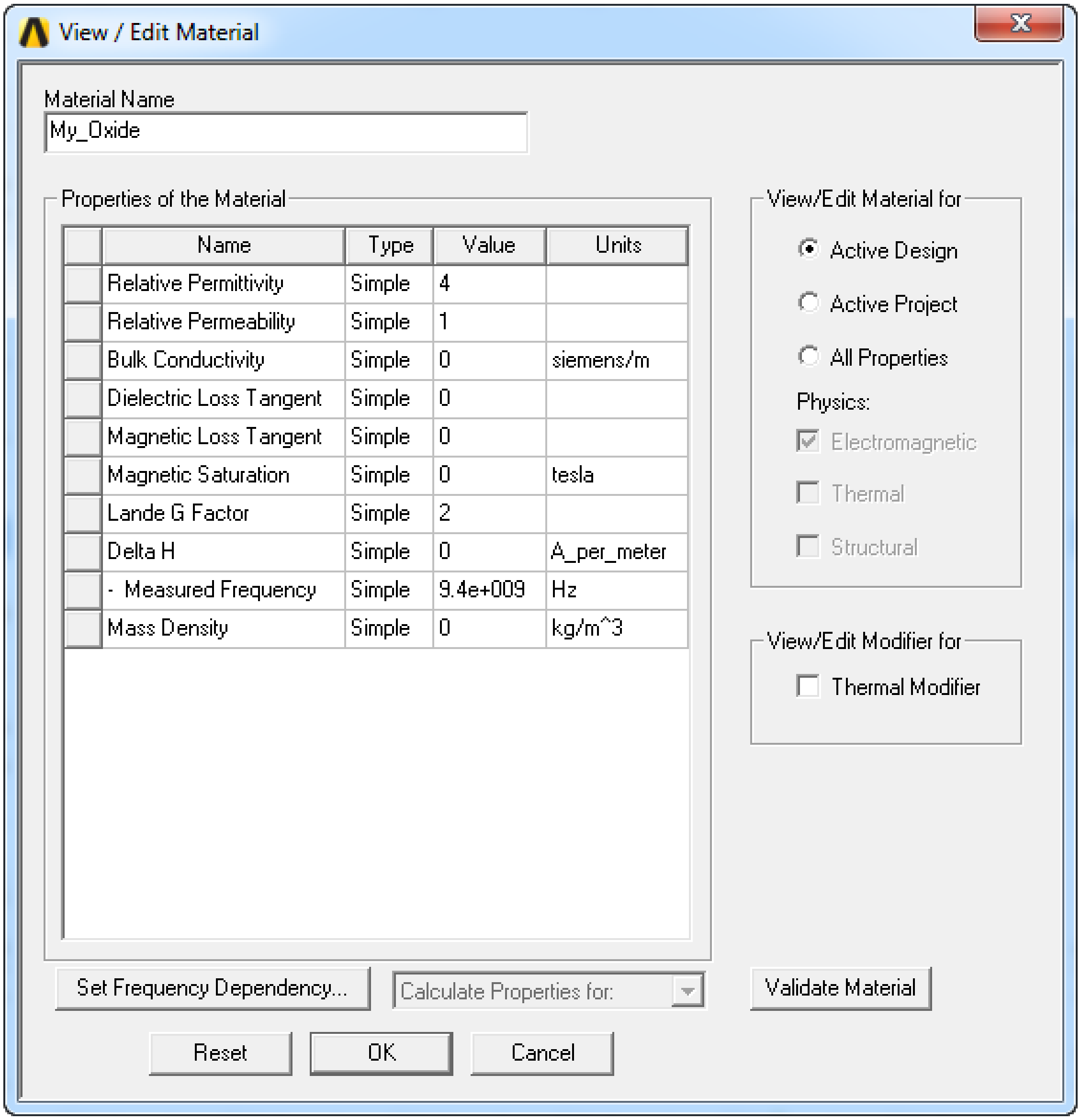


Figure 3-5 View/Edit Material dialog

5. Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog boxes to exit.

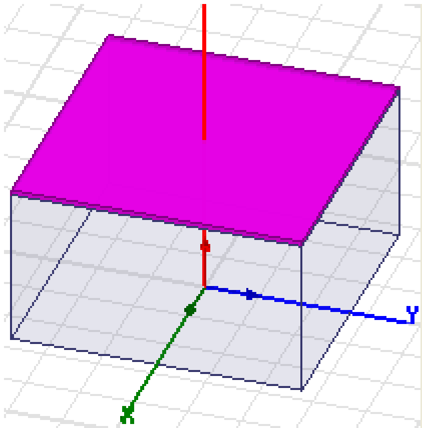


Figure 3-6 Oxide Substrate

Create Passivation

To create passivation, draw a box and specify its size and location as follows:

1. Draw the box freehand.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	Global		
Position	-270 ,-270 ,309.8	um	-270um , -270um , 309.8...
XSize	540	um	540um
YSize	540	um	540um
ZSize	0.7	um	0.7um

Figure 3-7 Command tab for passivation

2. On the **Command** tab edit the fields as shown in "Command tab for passivation" above and on the **Attribute** tab rewrite the **Name** field to *Pass*.
3. Select **Edit** from the **Materials** drop down menu.
4. Click **Add Material** and edit the fields as in the following figure.

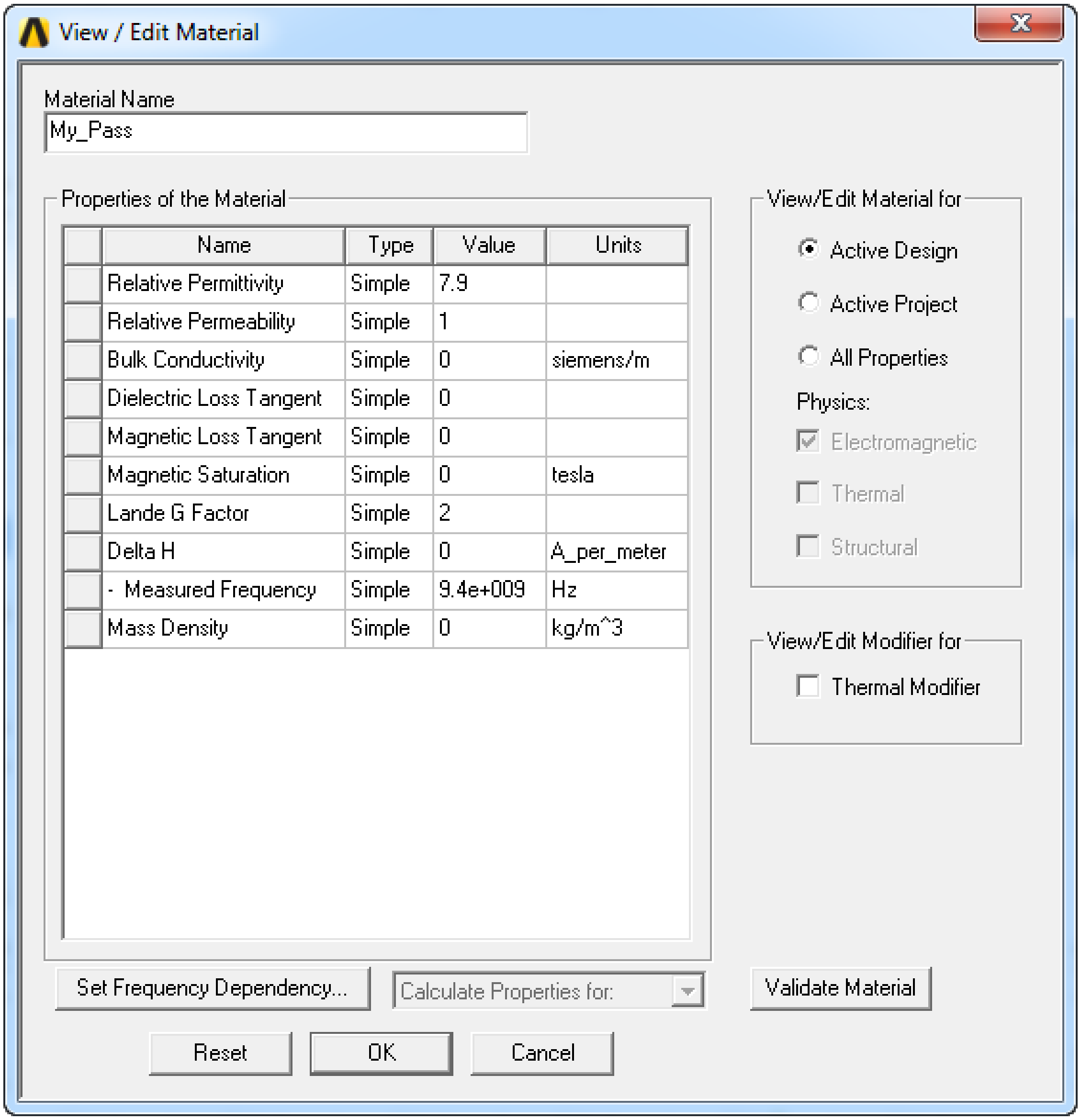


Figure 3-8 View/Edit Material dialog

5. Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog boxes to exit.

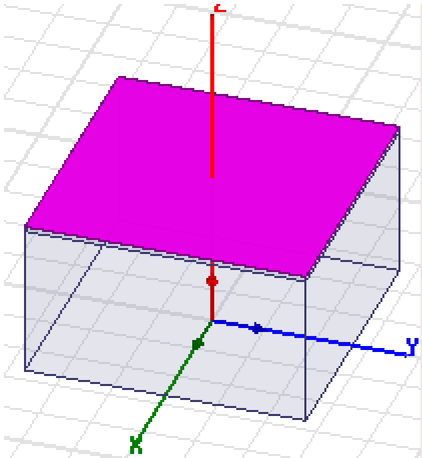


Figure 3-9 Passivation applied

Create Air Body

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

1. Draw a box freehand.
2. Set the **Command** dialog box as in the following figure.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	Global		
Position	-270 ,-270 ,0	um	-270um , -270um , 0um
XSize	540	um	540um
YSize	540	um	540um
ZSize	600	um	600um

Figure 3-10 Properties dialog for Air

3. On the **Attribute** tab, rename object to *Air.*
4. Ensure that the **Material** selected is *vacuum* and click **OK**.

   The box gets updated with the new dimensions and properties that you set.

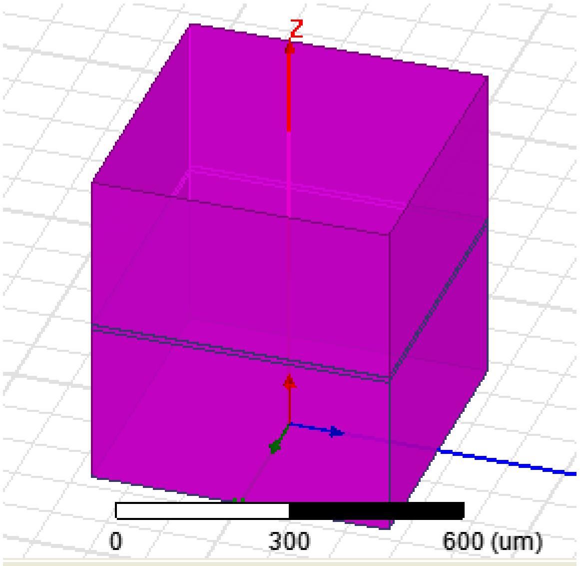


Figure 3-11 Air Enclosure drawn

Assign Radiation Boundary

Now assign the radiation boundary to the air box.

1. Select **Air** from the **History Tree** dialog box**.**

   The air body gets highlighted.

2. Right click **Air** and select **Assign Boundary>Radiation** from the short-cut menu.

   The **Radiation Boundary** dialog box appears.

3. Edit the fields as shown in the figure below and click **OK**.

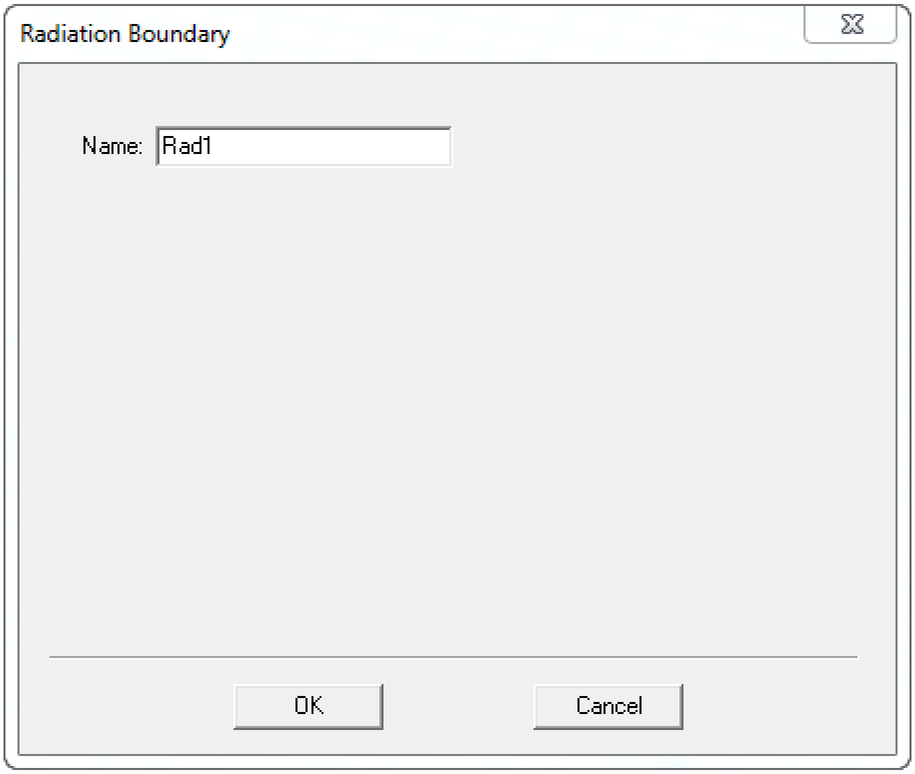


Figure 3-12 Radiation Boundary

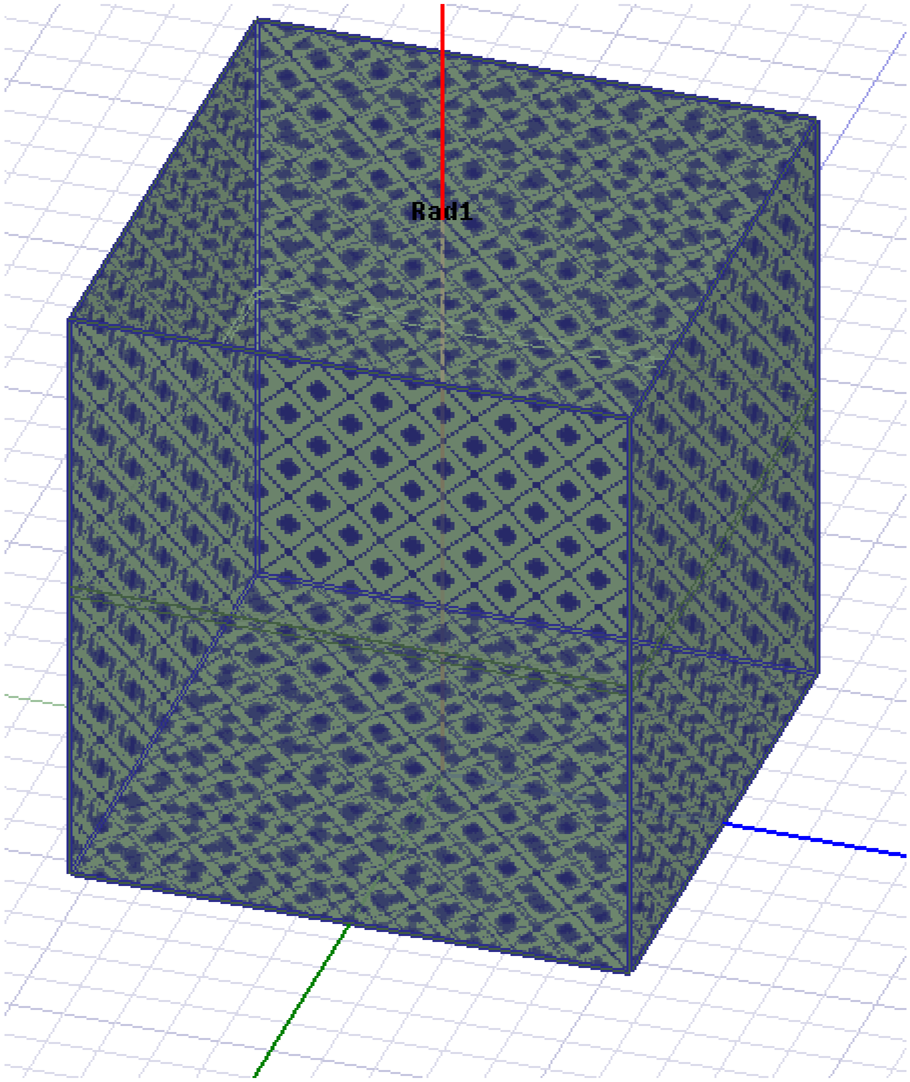


Figure 3-13

Create Ground

To create the ground, draw a rectangle as described below.

1. Click **Draw**> **Rectangle**.
2. Draw a rectangle freehand.

   The **Properties** dialog box appears.
3. Click **OK** to accept the values in the **Properties** dialog box.
4. Double-click **CreateRectangle** from the history tree.

   The **Command** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate System	Global		
Position	-270 ,-270 ,0	um	-270um , -270um , 0um
Axis	Z		
XSize	540	um	540um
YSize	540	um	540um

Figure 3-14 Properties for Rectangle

5. Edit the fields as shown in "Properties for Rectangle" above.

   The rectangle updates itself with the new settings.

6. Double-click **Rectangle1** in the history tree and on the **Attribute** dialog box enter *Ground* in the **Name** field and click **OK**.

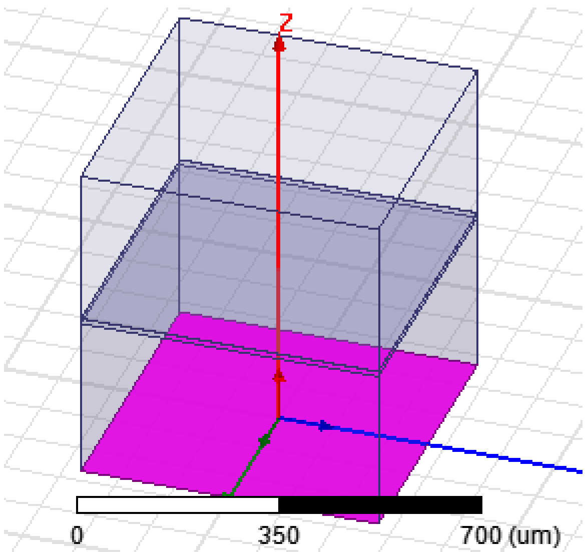


Figure 3-15 The structure with rectangle drawn

Assign Perfect E Boundary to the Ground

1. Click outside the structure to deselect all 2D and 3D objects.
2. Select **Ground** from the **history tree to highlight it**.
3. Right-click **Ground** and select **Assign Boundary**>**Perfect E** from the short-cut menu.

   The **Perfect E Boundary** dialog box appears.

4. Enter *PerfE_Ground* in the **Name** field.

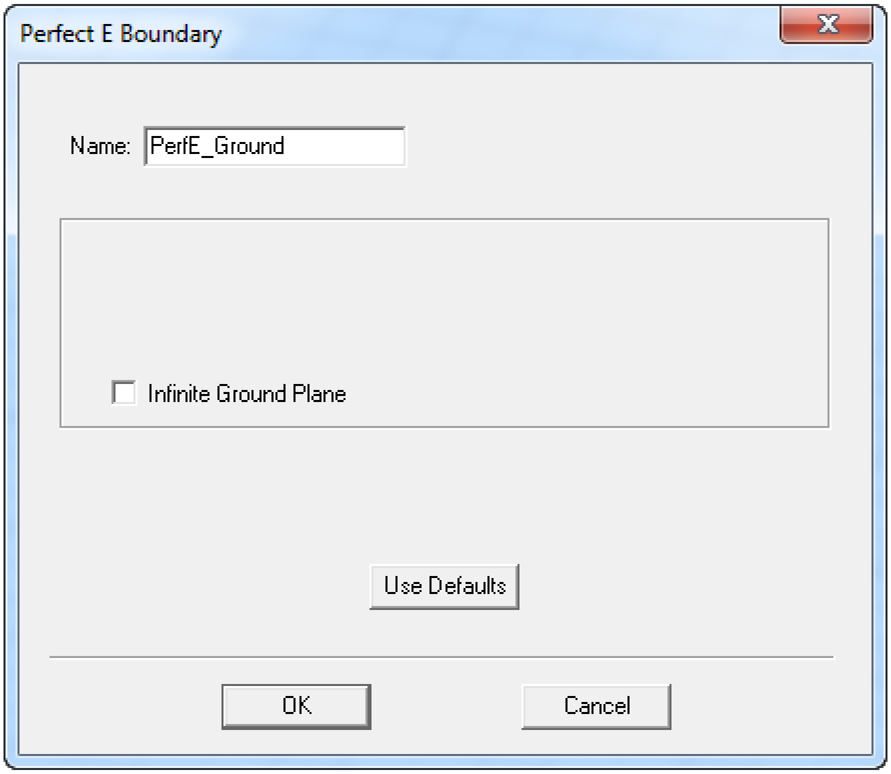


Figure 3-16 Perfect E boundary dialog

5. Leave the **Infinite Ground Plane** unchecked and click **OK**.

   The Perfect E boundary is applied and the Message Manager gives the following warning:

   - *Boundary Rad1 and Boundary PerfE_Ground overlap*.

   This is because you applied the PerfE boundary on a face that already has the Radiation boundary. However, PerfE_Ground overrides the Radiation boundary on that face owing to a higher priority.

   Note By default priority is assigned according to the order in which the boundaries are applied. Since PerfE_Ground was assigned after Rad1, it has a higher priority. HFSS lets you re-prioritize the boundaries, but it is not needed for this project.

Hide Dielectrics

1. Click **Edit>Select All Visible**
2. Click **View>Visibility>Hide Selection>All Views**

   All the objects are now hidden.

Create Spiral Inductor Geometry

Before you create the spiral inductor, set the default material.

1. From the **Modeler Materials** toolbar, choose **Select.**

   The **Select Definition** dialog box appears.

2. Click **Add Material**.

   The **View/Edit Material** dialog box appears.

   Edit the fields as shown in the following figure.

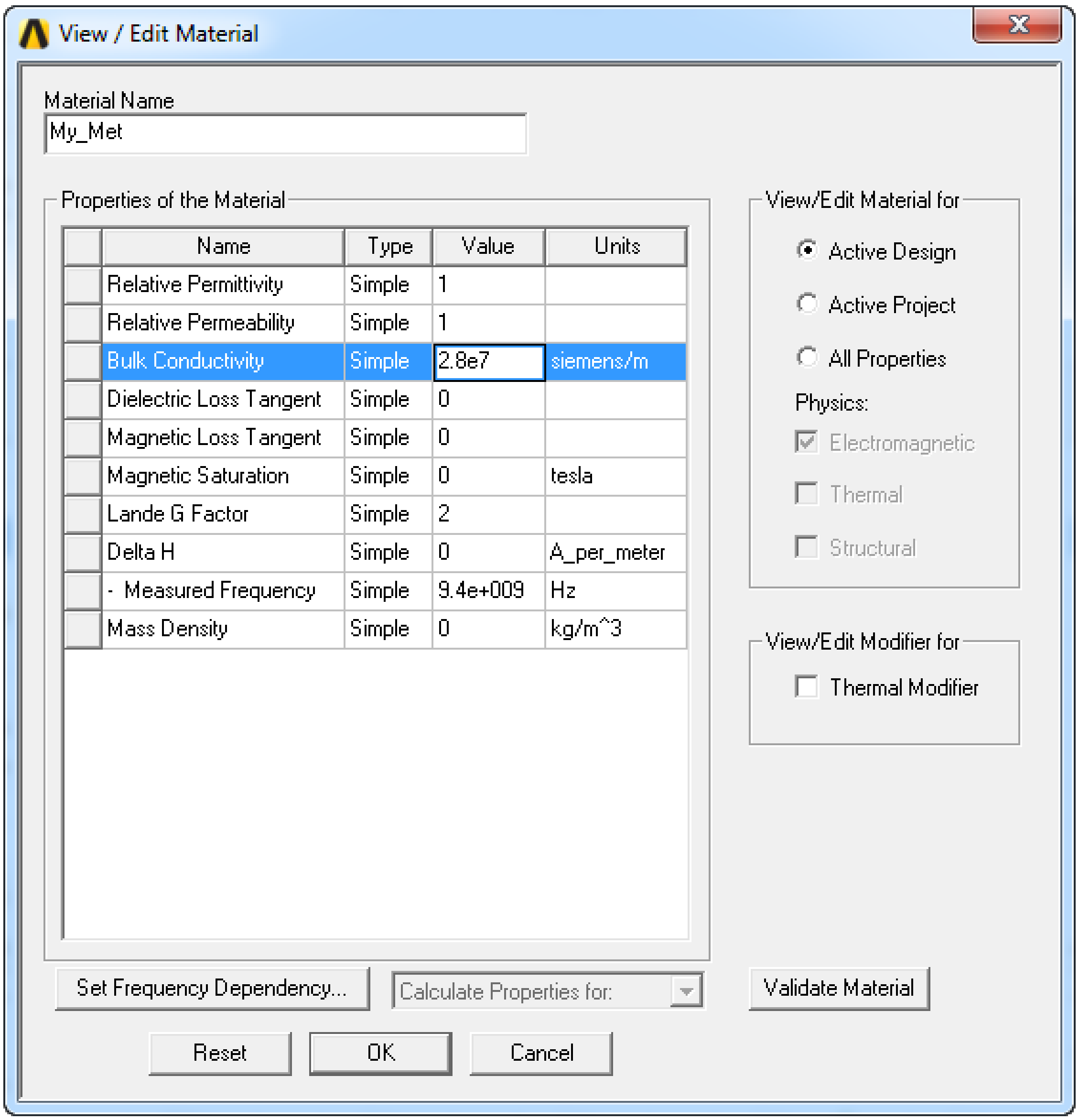


Figure 3-17 View/Edit Material dialog

3. Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog box to exit.

Create Offset Coordinate System

1. Click **Modeler>Coordinate System>Create>Relative CS>Offset**
2. In the coordinate fields, enter the origin as follows:

   X:*0.0*, **Y:** *0.0*, **Z:***304.8*.

   Note The co-ordinate fields appear on the status bar at the bottom and are titled **Select the origin**.

3. Press **Enter.**

Create Spiral Path

To create the spiral draw a Polyline using one of the following methods:

- Create Polyline Freehand
- Create Polyline From the Status Bar

Create Polyline Freehand

This section shows how to create the spiral by drawing it free hand and then, editing its coordinates.

1. Click **Draw>Line**.
2. Click anywhere in the modeler to establish the first point and drag the cursor to draw a line and click again, to establish the second point as shown in the figure below.

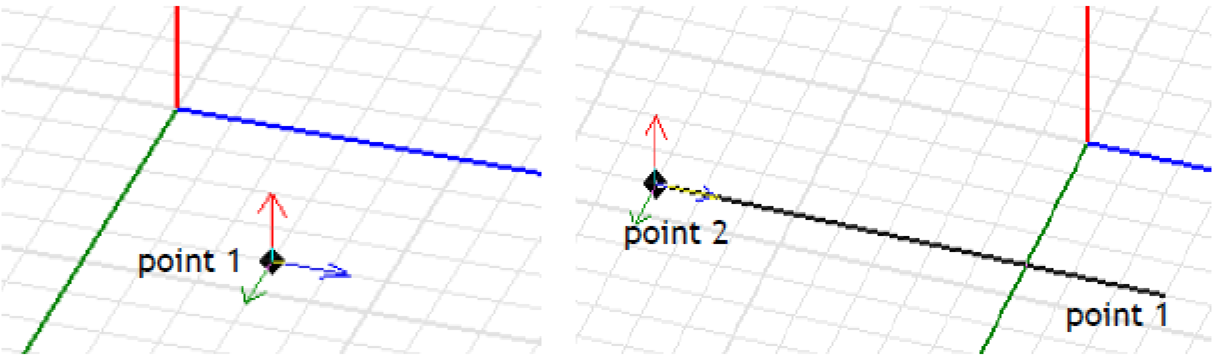


Figure 3-18

3. Continue to drag and click the mouse to establish 13 such points as shown in the figure below where we have labeled all the 13 points.
4. Right click and select **Done** from the short cut menu.

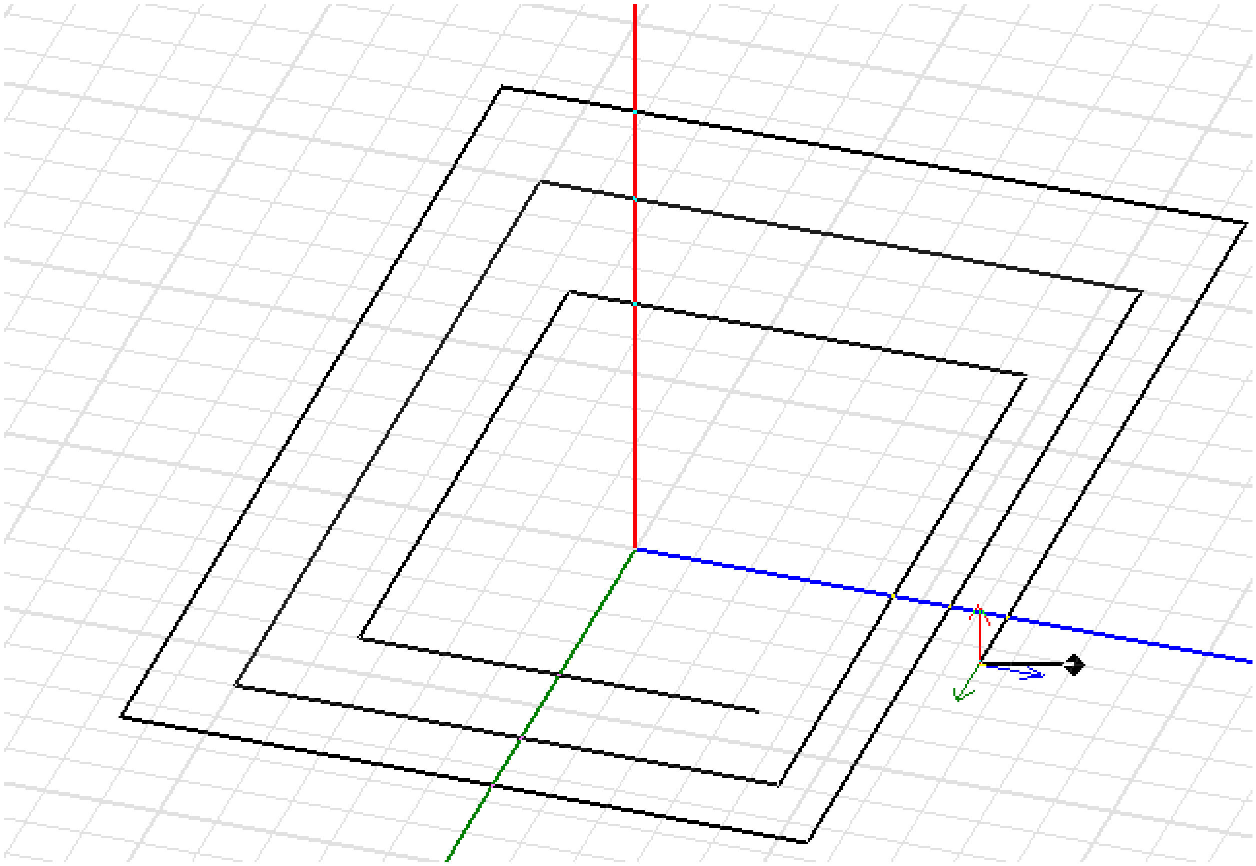


Figure 3-19 Polyline with 13 points

5. Double-click **CreateLine** from the history tree and edit the coordinates for the vertices in its Segment tab as shown below.

Line 1

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-67.5 ,7.5 ,1	um	-67.5um , 7.5um , 1um
Point2	-67.5 ,-67.5 ,1	um	-67.5um , -67.5um , 1um

Figure 3-20 Coordinates for Line1

6. Edit the fields for each of the 12 **CreateLine** options on their corresponding **Segment** tabs as shown below.

Line 2

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-67.5 ,-67.5 ,1	um	-67.5um , -67.5um , 1um
Point2	84 ,-67.5 ,1	um	84um , -67.5um , 1um

Line 3

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	84 ,-67.5 ,1	um	84um , -67.5um , 1um
Point2	84 ,84 ,1	um	84um , 84um , 1um

Line 4

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	84 ,84 ,1	um	84um , 84um , 1um
Point2	-84 ,84 ,1	um	-84um , 84um , 1um

Line 5

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-84 ,84 ,1	um	-84um , 84um , 1um
Point2	-84 ,-84 ,1	um	-84um , -84um , 1um

Line 6

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-84 ,-84 ,1	um	-84um , -84um , 1um
Point2	100.5 ,-84 ,1	um	100.5um , -84um , 1um

Line 7

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	100.5 ,-84 ,1	um	100.5um , -84um , 1um
Point2	100.5 ,100.5 ,1	um	100.5um , 100.5um , 1um

Figure 3-21 Coordinates for Line 2 through Line 7

Line 8

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	100.5 ,100.5 ,1	um	100.5um , 100.5um , 1um
Point2	-100.5 ,100.5 ,1	um	-100.5um , 100.5um , 1um

Line 9

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-100.5 ,100.5 ,1	um	-100.5um , 100.5um , 1um
Point2	-100.5 ,-100.5 ,1	um	-100.5um , -100.5um , 1um

Line 10

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	-100.5 ,-100.5 ,1	um	-100.5um , -100.5um , 1um
Point2	117 ,-100.5 ,1	um	117um , -100.5um , 1um

Line 11

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	117 ,-100.5 ,1	um	117um , -100.5um , 1um
Point2	117 ,0 ,1	um	117um , 0um , 1um

Line 12

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	117 ,0 ,1	um	117um , 0um , 1um
Point2	131 ,0 ,1	um	131um , 0um , 1um

Figure 3-22 Coordinates for Line 8 through Line 12

Create Polyline from the Status Bar

In this method, enter the coordinates of the points in the status bar as described below.

1. Click **Draw>Line.**
2. Edit the coordinate entry fields as follows:

   Enter the vertex field for point 1:

X: **-67.5**, Y: **7.5**, Z: **1.0** Press the **Enter** key.

Enter the vertex field for point 2:

X: **-67.5**, Y: **-67.5**, Z: **1.0** Press the **Enter** key.

Enter the vertex field for point 3:

X: **84.0**, Y: **- 67.5**, Z: **1.0** Press the **Enter** key.

Enter the vertex field for point 4:

X: **84.0**, Y: **84.0**, Z: **1.0** Press the **Enter** key.

Enter the vertex field for point 5:

X: - **84.0**, Y: **84.0**, Z: **1.0** Press the **Enter** key.

Enter the vertex field for point 6:

X: - **84.0**, Y: **-84.0,** Z: **1.0**, Press the **Enter** key.

Enter the vertex field for point 7:

X: **100.5**, Y: **-84.0**, Z: **1.0,** Press the **Enter** key.

Enter the vertex field for point 8:

X: **100.5**, Y: **100.5**, Z: **1.0**, Press the **Enter** key.

Enter the vertex field for point 9:

X: - **100.5**, Y: **100.5**, Z: **1.0,** Press the **Enter** key.

Enter the vertex field for point 10:

X: - **100.5**, Y: **- 100.5**, Z: **1.0**, Press the **Enter** key.

Enter the vertex field for point 11:

X: **117.0**, Y: **-100.5**, Z: **1.0**, Press the **Enter** key.

Enter the vertex field for point 12:

X: **117.0**, Y: **0.0**, Z: **1.0**, Press the **Enter** key.

Enter the vertex field for point 13:

X: **131.0**, Y: **0.0**, Z: **1.0**, Press the **Enter** key.

3. Using the mouse, right-click and from the short-cut menu select **Done**.
4. Select **Attribute** and enter *Spiral* in the **Name** field and click **OK.**

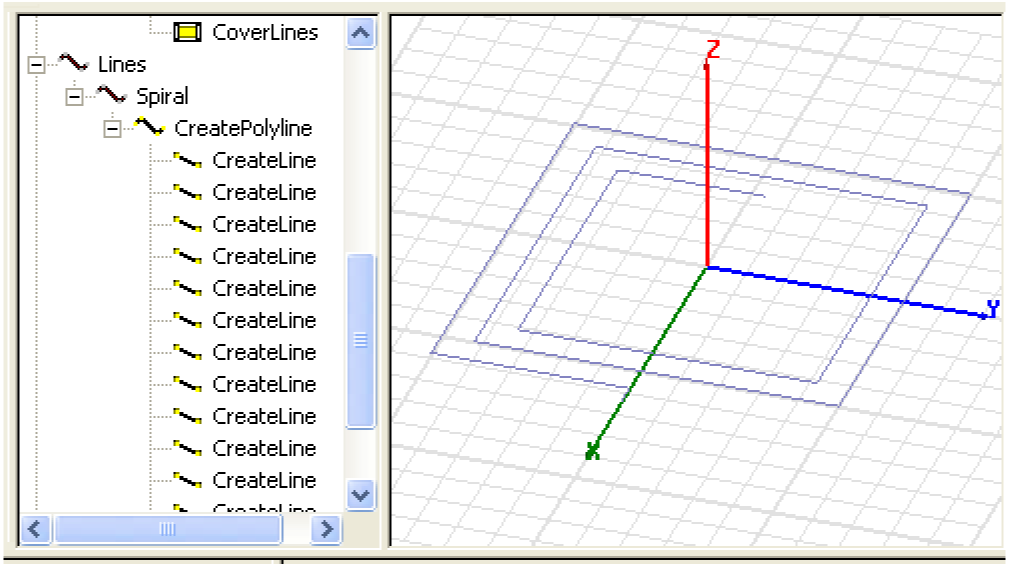


Figure 3-23 Spiral drawn

Assign Thickness to the Spiral

To assign trace width and thickness perform the following:

1. Right-click **Create Polyline** under **Spiral** from the **History Tree**.
2. Select **Properties** from the short-cut menu.

   The **Properties** dialog box appears.

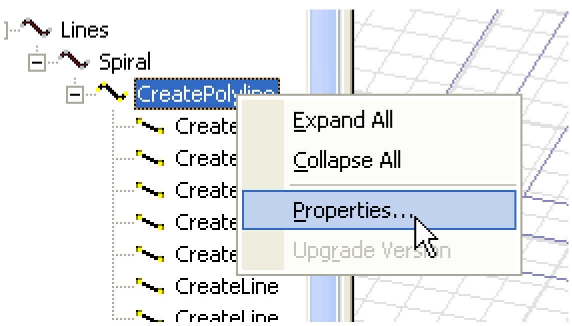


Figure 3-24 History Tree

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
Top Width	0	um	0um
Height	2	um	2um
Number of Segments	0		0
Bend Type	Corner		

Figure 3-25 Command dialog box

3. Edit the fields as shown in "Command dialog box" above. and click **OK** to close the **Properties** dialog box.

   The spiral is assigned the thickness that you set.

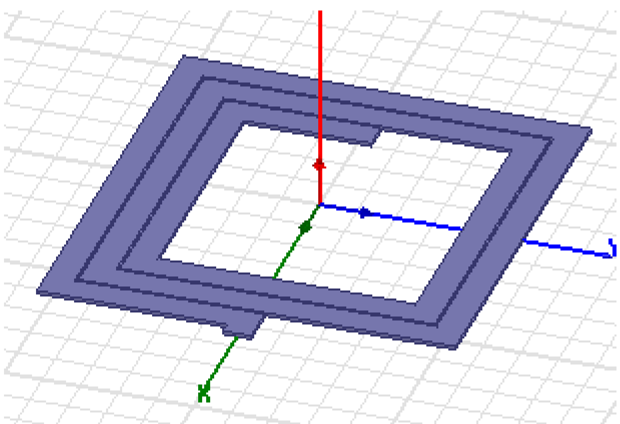


Figure 3-26 The updated spiral

Create Underpass

Before creating the underpass ensure the grid plane is *XY* as follows:

1. Click **Modeler>Grid Plane>XY.**

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-60 ,7.5 ,-0.8	um	-60um , 7.5um , -0.8um
XSize	-75	um	-75um
YSize	-15	um	-15um
ZSize	-0.5	um	-0.5um

Figure 3-27 Properties dialog (underpass)

2. Draw a box freehand and edit the fields on the **Command** tab as shown in "Properties dialog (underpass)" above. .
3. On the **Attribute** tab rename object as *Underpass* and click **OK** to close the **Properties** dialog box.

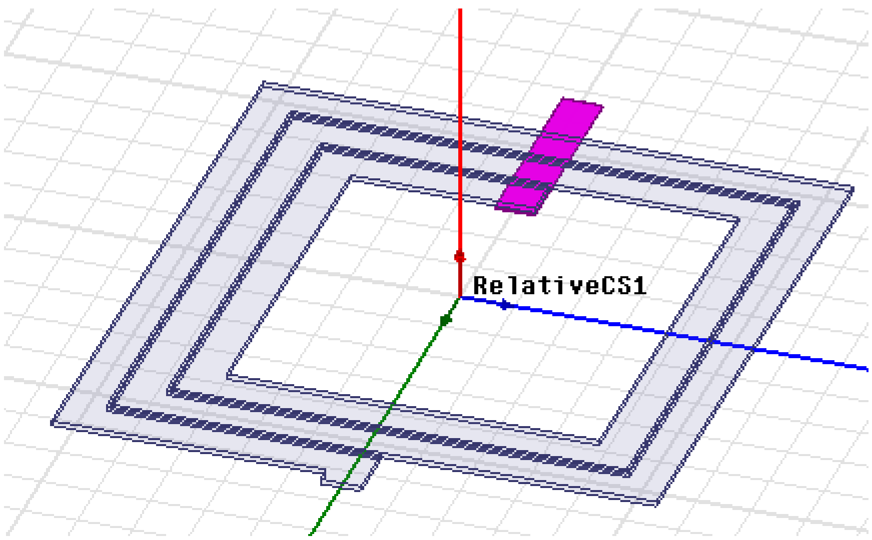


Figure 3-28 The Underpass

Create Via1

To create the Via, first draw a box.

1. Draw a box freehand and on the **Command** tab edit the fields as shown in "Properties dialog Via1" on the facing page. .
2. On the **Attribute** tab enter *Via1* in the **Name** field and click **OK**.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-60 ,7.5 ,0	um	-60um , 7.5um , 0um
XSize	-15	um	-15um
YSize	-15	um	-15um
ZSize	-0.8	um	-0.8um

Figure 3-29 Properties dialog Via1

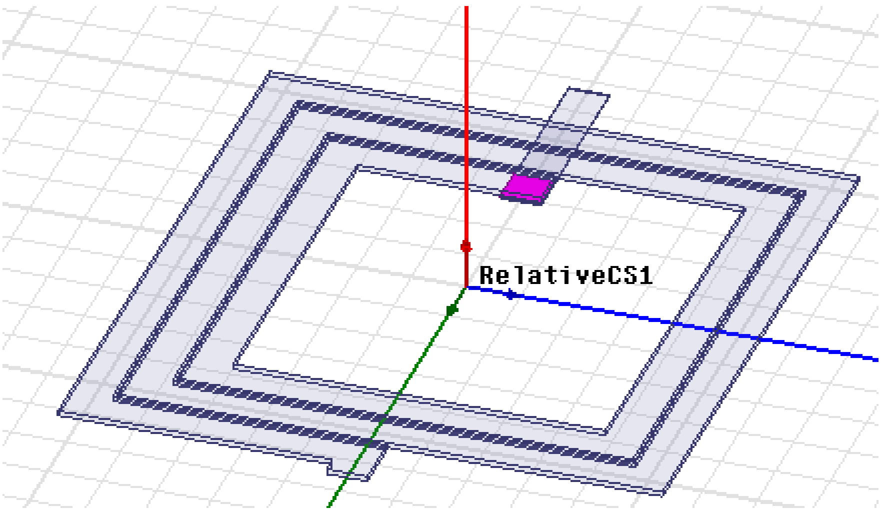


Figure 3-30 Via1 applied

Create Via2

To create another via, again, draw a box.

1. **Draw a box freehand and edit the fields on the Command tab as shown in** "Via2 Properties" below. .

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-120 ,7.5 ,0	um	-120um , 7.5um , 0um
XSize	-15	um	-15um
YSize	-15	um	-15um
ZSize	-0.8	um	-0.8um

Figure 3-31 Via2 Properties

2. On the **Attribute** tab enter *Via2* in the **Name** field and click **OK**.

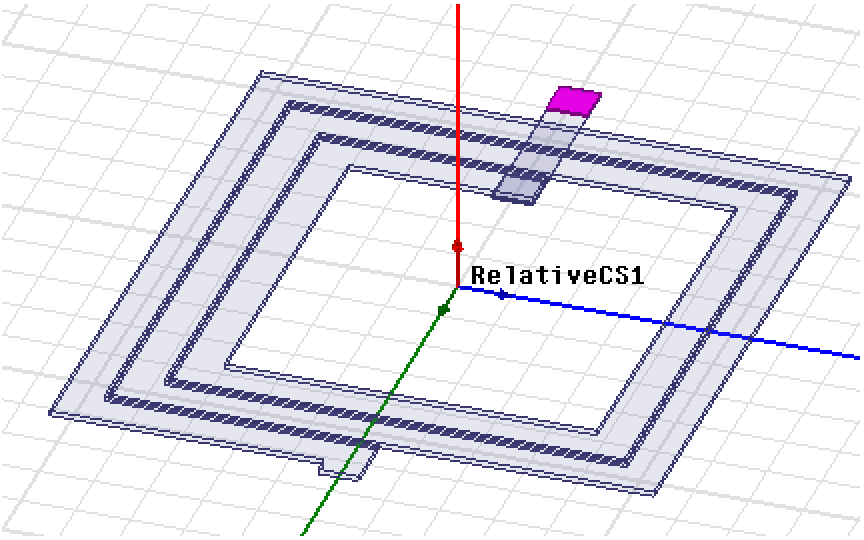


Figure 3-32 Via2 applied

Create Feed

1. Draw a box freehand.

   The **Properties** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-120 ,7.5 ,0	um	-120um , 7.5um , 0um
XSize	-22	um	-22um
YSize	-15	um	-15um
ZSize	2	um	2um

Figure 3-33 Feed Properties

2. Edit the fields as shown in "Feed Properties" above. and on the **Attribute** tab enter *Feed* in the **Name** field and click **OK** to close the **Properties** dialog box.
3. Do **Ctrl+D** to fit the view.

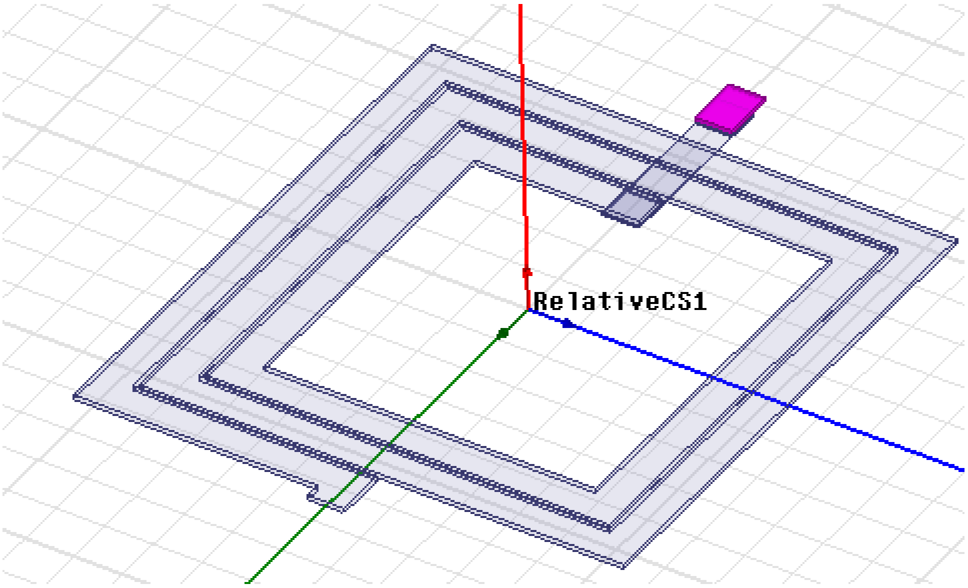


Figure 3-34 Feed applied

Unite Spiral Objects

You will now unite the spiral objects.

1. Click **Spiral,** press the Ctrl key and select **Via1, Via2, Feed,** and **Underpass.**

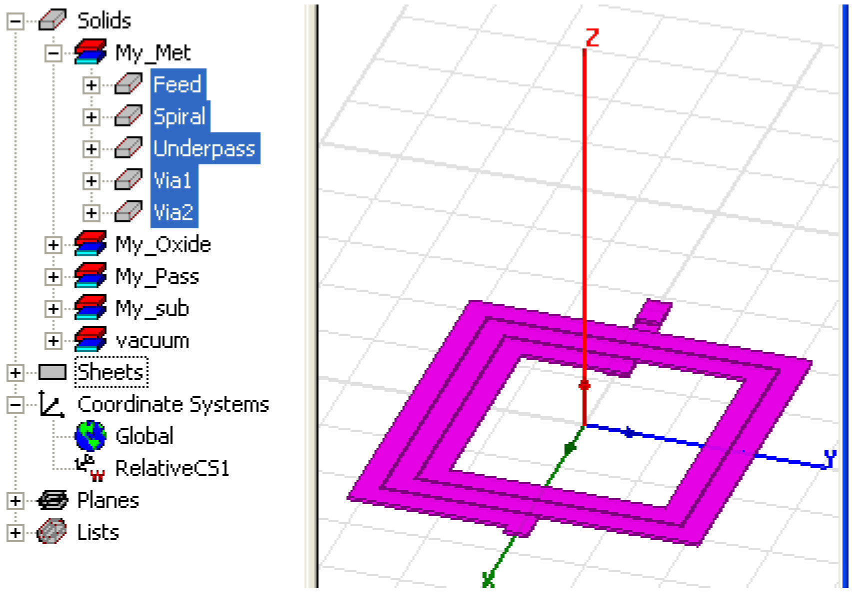


Figure 3-35 The pieces united

Note The order in which you select the objects determines the name of the united structure. For example if you select **spiral** first followed by the rest, the united structure will be named spiral. If you select **Feed** first, then, the united structure will be named as **Feed**.

2. Click **Modeler>Boolean>Unite**

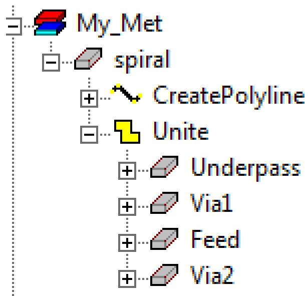


Figure 3-36 The united object names itself as spiral

3. Do **Ctrl+D** to fit the view.
4. Double-click **spiral from the history tree and make sure Solve Inside is unchecked on the Attribute tab.**

   Note The conductive material is represented by a boundary condition that removes the need to solve inside metal.

Seed Mesh Conductors

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. Click **Edit>Select All Visible** and select Spiral.
2. Click **HFSS> Mesh Operations>Assign>Inside Selection>Length Based**

   The Element Length Based Refinement dialog box appears.
3. Edit the fields as shown in the figure below and click **OK**.

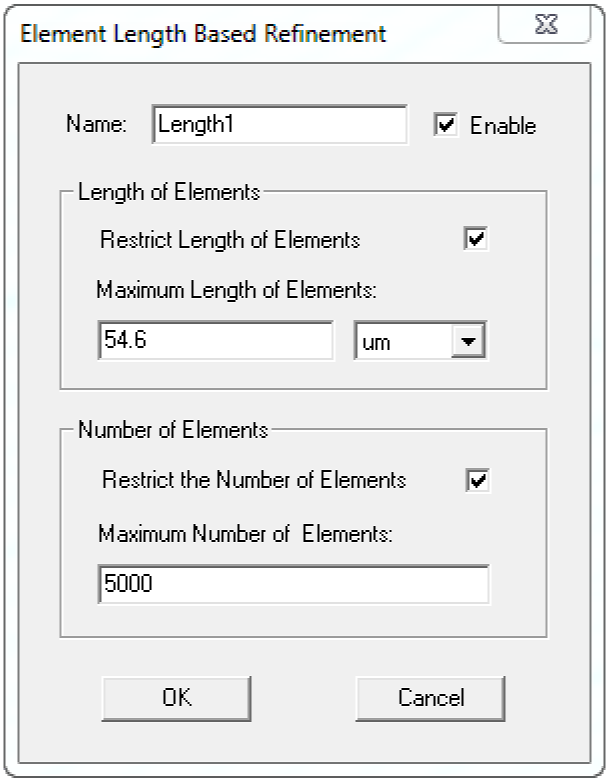


Figure 3-37 Element Length Based Refinement settings

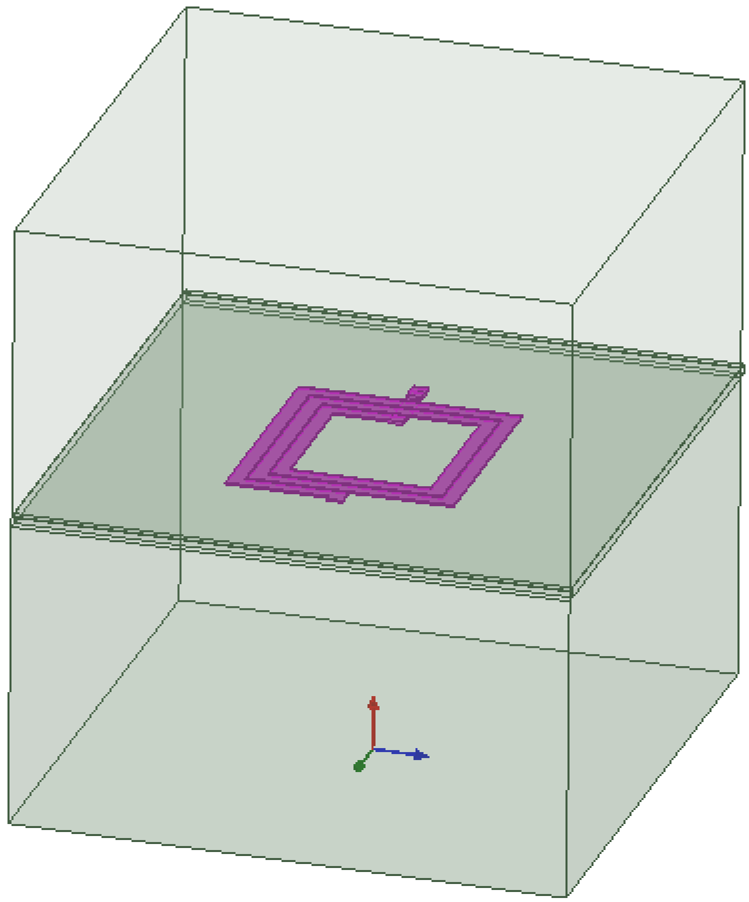


Figure 3-38

Create Ground Ring

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

1. Click **Draw**>**Box**.
2. Draw a box freehand.

   The **Properties** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-225 ,-225 ,0	um	-225um , -225um , 0um
XSize	450	um	450um
YSize	450	um	450um
ZSize	2	um	2um

Figure 3-39 Ring Properties

3. On the **Command** tab edit the fields as shown in "Ring Properties" above.
4. On the **Attribute** tab enter *Ring* in the **Name** field and select **Edit** from the **Materials** drop-down menu.

   The **Select Definition** window appears.
5. Type *pec* in the **Search by Name** field.
6. Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog boxes to exit.

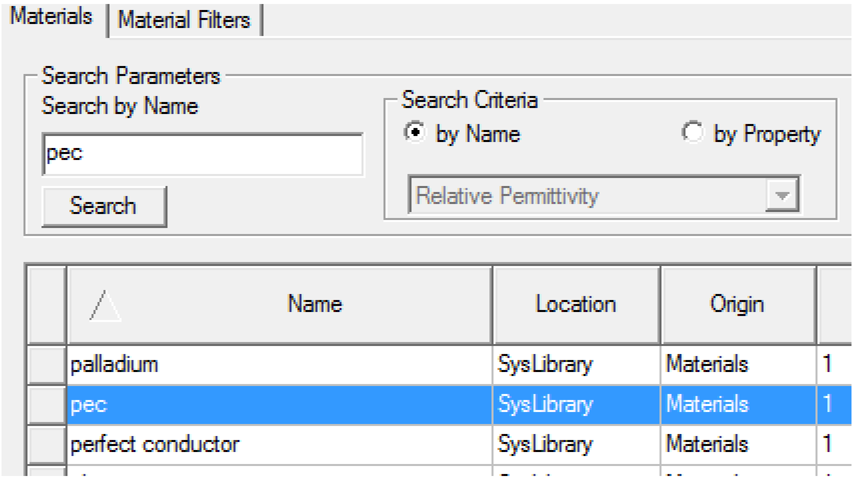


Figure 3-40 Select Definition window

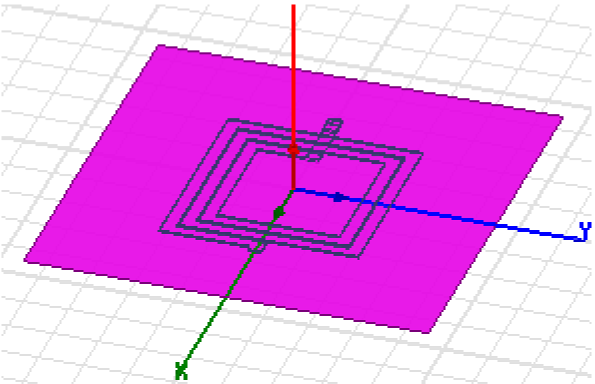


Figure 3-41 Ring applied

Create Inner Ring

To create the inner ring, again draw a box.

1. Click **Draw>Box**.
2. Draw a box freehand.

   The **Properties** dialog box appears.
3. On the **Command** tab, edit the fields as shown in "The Properties dialog box for Inner Ring" below. .

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-210 ,-210 ,0	um	-210um , -210um , 0um
XSize	420	um	420um
YSize	420	um	420um
ZSize	2	um	2um

Figure 3-42 The Properties dialog box for Inner Ring

4. On the **Attribute** tab enter *Inner* in the **Name** field and ensure that the **Material** assigned is *pec* and click **OK**.

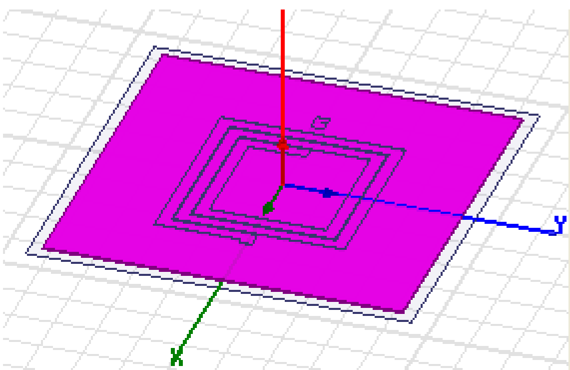


Figure 3-43 Inner ring drawn

Complete the Ring

1. Click **Edit>Select>By Name**

   The Select Object dialog box appears.

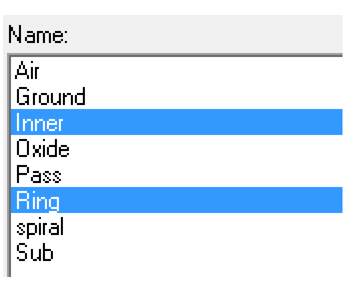


Figure 3-44 Select Object dialog box

2. Select **Ring,** press the **Ctrl** key and click **Inner** and click **OK.**
3. Click **Modeler>Boolean>Subtract**

   The **Subtract** dialog box appears.
4. Verify *Ring* is in the **Blank Parts** and *Inner* in the **Tool Parts** and click **OK**.

   Post subtraction, the structure should resemble the one in "The subtracted ring" below. .

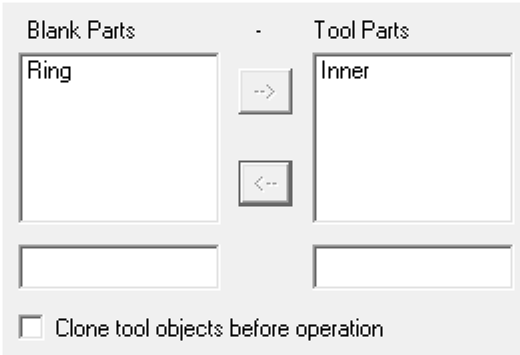


Figure 3-45 Subtract dialog box

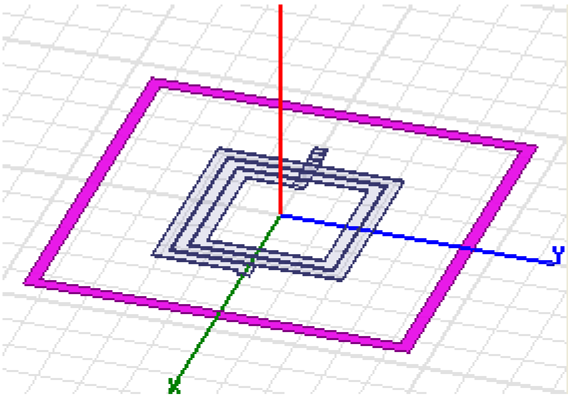


Figure 3-46 The subtracted ring

Create Extension 1

1. Draw a box freehand.

   The **Properties** dialog box appears.
2. On the **Command** tab edit the fields as shown in "Extension1 properties" on the next page. .
3. On the **Attribute** tab enter the **Name** as *Ring_Ext1*, ensure that *pec* is selected from the **Material** drop-down and click **OK.**

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	-157 ,7.5 ,0	um	-157um , 7.5um , 0um
XSize	-53	um	-53um
YSize	-15	um	-15um
ZSize	2	um	2um

Figure 3-47 Extension1 properties

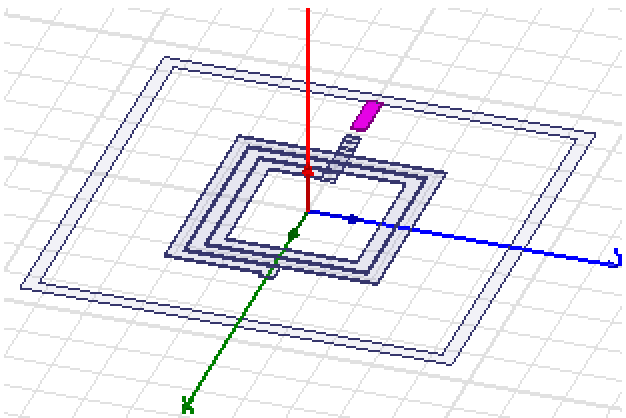


Figure 3-48 The Ring_Ext1 applied

Create Extension 2

1. Draw a box freehand.

   The **Properties** dialog box appears.

2. Edit the fields as shown in "Properties dialog box for Ring_Ext2" below. .

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	RelativeCS1		
Position	146 ,7.5 ,0	um	146um , 7.5um , 0um
XSize	64	um	64um
YSize	-15	um	-15um
ZSize	2	um	2um

Figure 3-49 Properties dialog box for Ring_Ext2

3. On the **Attribute** tab enter the **Name** as *Ring_Ext2* and click **OK**.

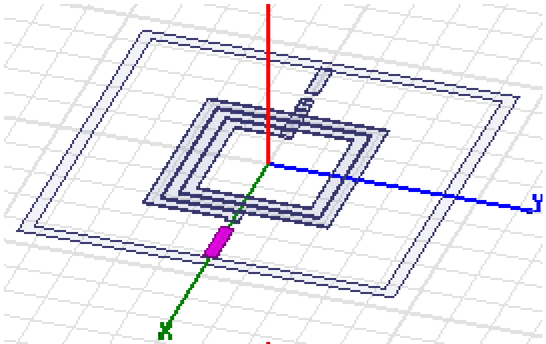


Figure 3-50 Ring_Ext2 applied

Create Source 1

1. Draw the rectangle freehand.

   The **Properties** dialog box appears.
2. Click **OK** to accept the current settings.
3. Double click **CreateRectangle** from the history tree.
4. Edit the fields in the **Command** dialog box as in the following figure.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate System	RelativeCS1		
Position	-142 ,7.5 ,1	um	-142um , 7.5um , 1um
Axis	Z		
XSize	-15	um	-15um
YSize	-15	um	-15um

Figure 3-51 Command dialog box for Source1

5. Click **Attribute** and enter **Name** type as *Source1* and click **OK**.

Create Source 2

1. Draw the rectangle freehand.

   The **Properties** dialog box appears.
2. Click **OK** to close the dialog box.
3. Under **Rectangle1**, double click **CreateRectangle** from the history tree.
4. Edit the fields as shown in the following figure.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate System	RelativeCS1		
Position	131 ,7.5 ,1	um	131um , 7.5um , 1um
Axis	Z		
XSize	15	um	15um
YSize	-15	um	-15um

Figure 3-52 Command dialog box for Source2

5. Double-click **Rectangle1** and enter *Source2* in the **Name** field and click **OK**.

Group the Conductors

1. Click **Edit>Select>By Name**
2. In the **Select Object** dialog box, select the **Ring, Ring_Ext1, Ring_Ext2**
3. Click **OK.**
4. Click **Modeler>Boolean>Unite**
5. Do **Ctrl+D** to fit the view.

Assign Excitation for Source1

We will use wave ports to excite source1:

1. Click **Source1 from the history tree.**

   Source1 gets highlighted in the structure.

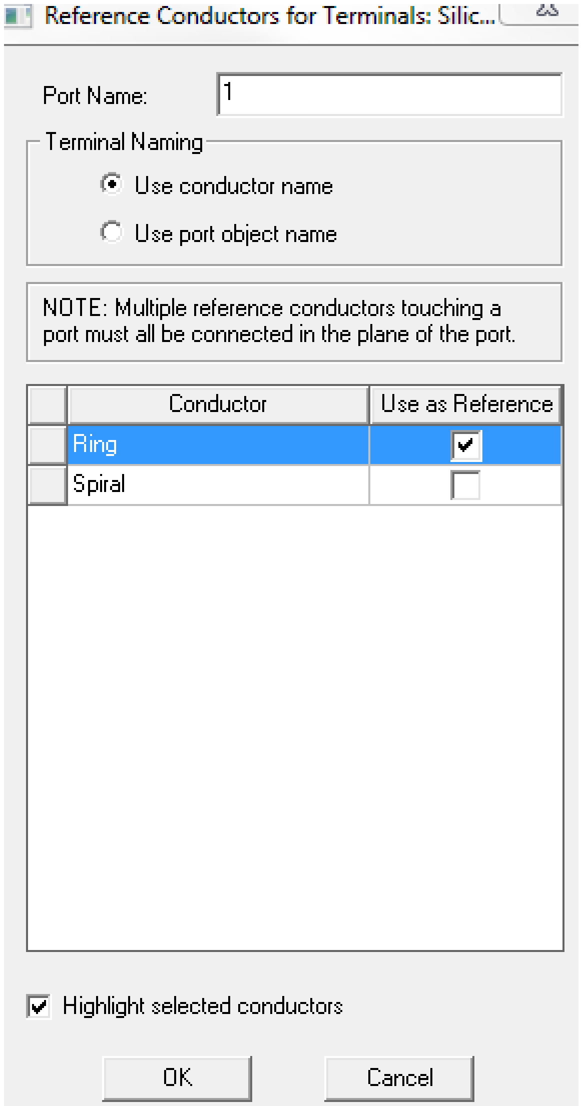


Figure 3-53 Reference Conductor Terminal dialog box

2. Right click **Source1**, and select **Assign Excitation>Assign>Lumped Port**
   Reference Conductor for Terminals dialog box appears.
3. Set the options as shown in "Reference Conductor Terminal dialog box" above. and click **OK**.

Assign Excitation for Source2

To select the object Source2:

1. In the **History** tree, expand the **Unassigned** objects tree.
2. Select **Source2.**

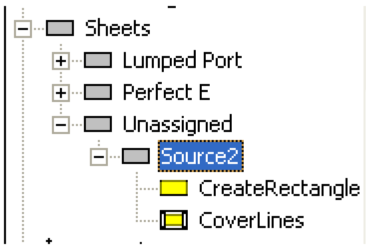


Figure 3-54 History Tree

To assign lumped port excitation

1. Click **HFSS>Excitations>Assign>Lumped Port**
2. Enter *2* for the Port Name.
3. Set the rest of the options as in "Reference Conductor Terminal dialog box" on the previous page. and click **OK**.

Show All

To show all objects do the following:

1. Click **View>Visibility>Show All>All Views**
2. Do **Ctrl+D** to fit the view.

Boundary Display (Optional)

Boundary display/solver view provides a snapshot of all boundaries in the model including ports and surface residing on the surrounding background object. It can be very useful for diagnosing problems with design setups.

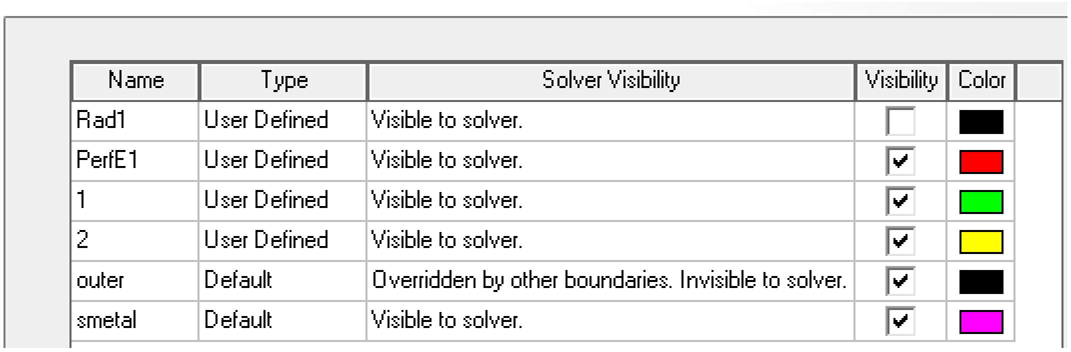
1. Click **HFSS>Boundary Display (Solver View)**

   The **Solver View of Boundaries** dialog box appears.

   Note HFSS identifies all the unique boundary conditions and ports to display where the boundaries are physically located in the model.

2. Select the boundaries you wish to view from the dialog box as shown in "Solver View of Boundaries dialog box" on the facing page. .

   The choices made here will show the boundaries in the **Modeler** field. See "Solver Boundaries selected" on the facing page. .



Name	Type	Solver Visibility	Visibility	Color
Rad1	User Defined	Visible to solver.	[ ]	
PerfE1	User Defined	Visible to solver.	[x]	
1	User Defined	Visible to solver.	[x]	
2	User Defined	Visible to solver.	[x]	
outer	Default	Overridden by other boundaries. Invisible to solver.	[x]	
smetal	Default	Visible to solver.	[x]	

Figure 3-55 Solver View of Boundaries dialog box

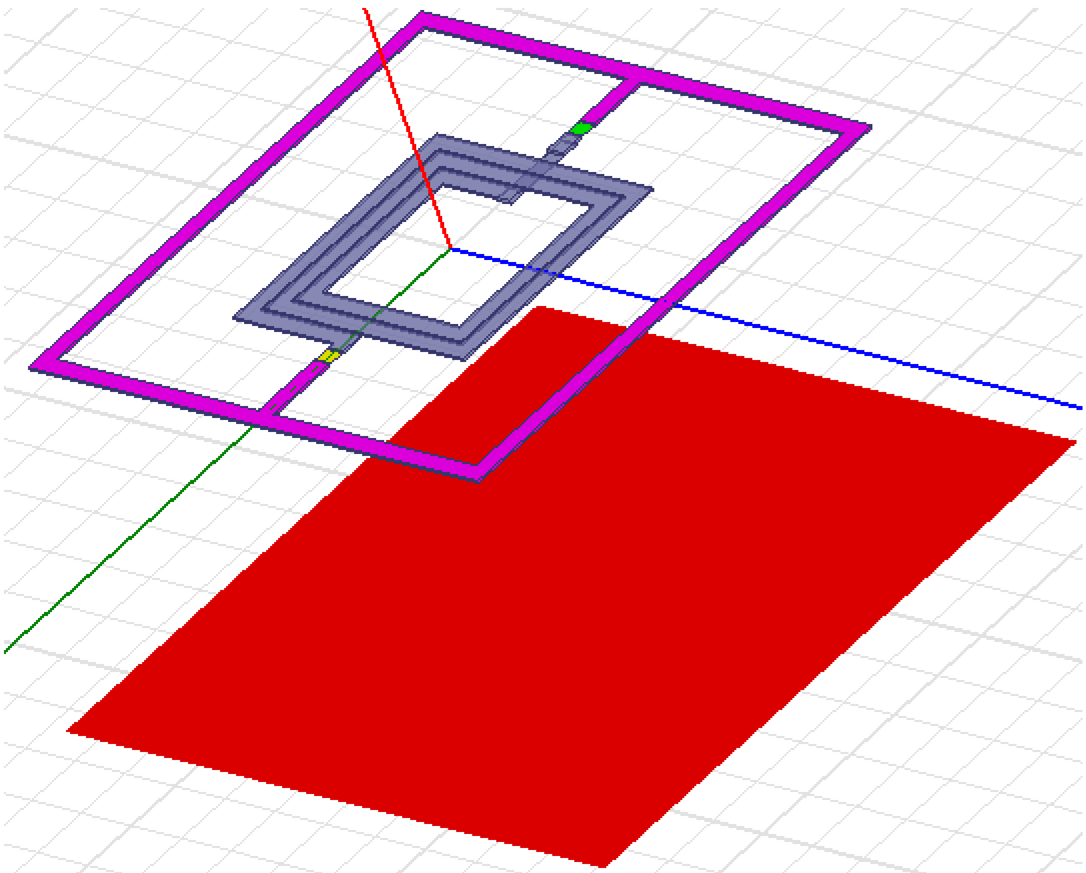


Figure 3-56 Solver Boundaries selected

Note If you double-click the fields under **Color**, you can change the color as you want from the palette that appears. The background is displayed as the outer boundary and the perfect conductors are displayed as the smetal boundary.

4 - Analyze Spiral Conductor

This chapter describes how to run the simulation and generate reports.

This chapter contains the following topics:

- Create Analysis Setup
- Add Frequency Sweep
- Model Validation
- Analyze the Model
- Solution Data
- Profile
- Convergence
- Matrix Data
- Mesh Statistics
- Generate Reports
- Create S-Parameter vs Frequency Plot
- Custom Equations – Output Variables
- Use Output Variables for Next Report

Create Analysis Setup

To create an analysis setup:

1. Click **HFSS>Analysis Setup>Add Solution Setup**
   The **Add Solution Set-up** dialog box appears.

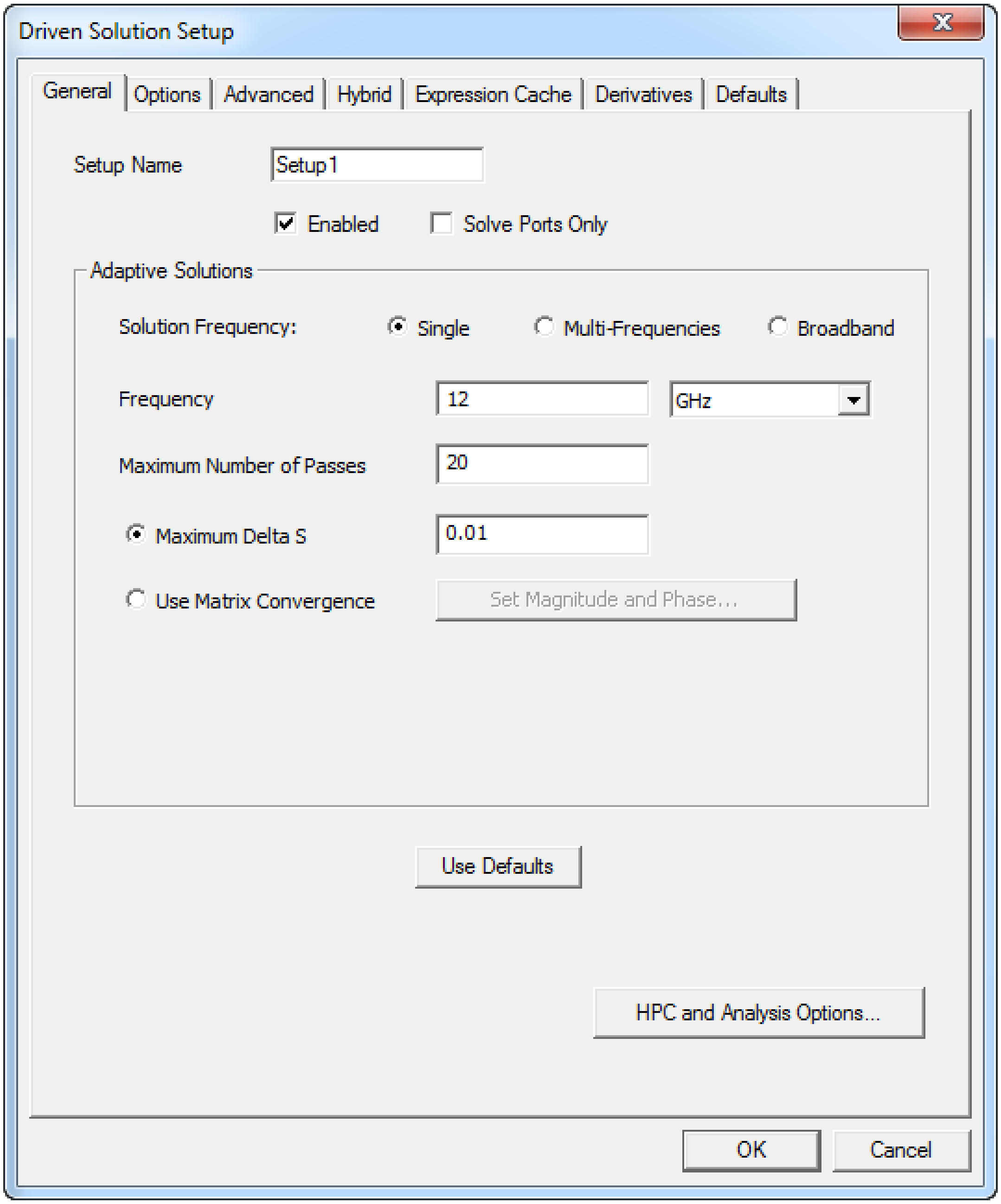


Figure 4-1 Solution Set-up window

2. In the **Solution Setup** window: click the **General** tab**.**
3. Edit the fields as shown in "Solution Set-up window" above.
4. Click **Options**, edit the fields as in the figure below and click **OK**.

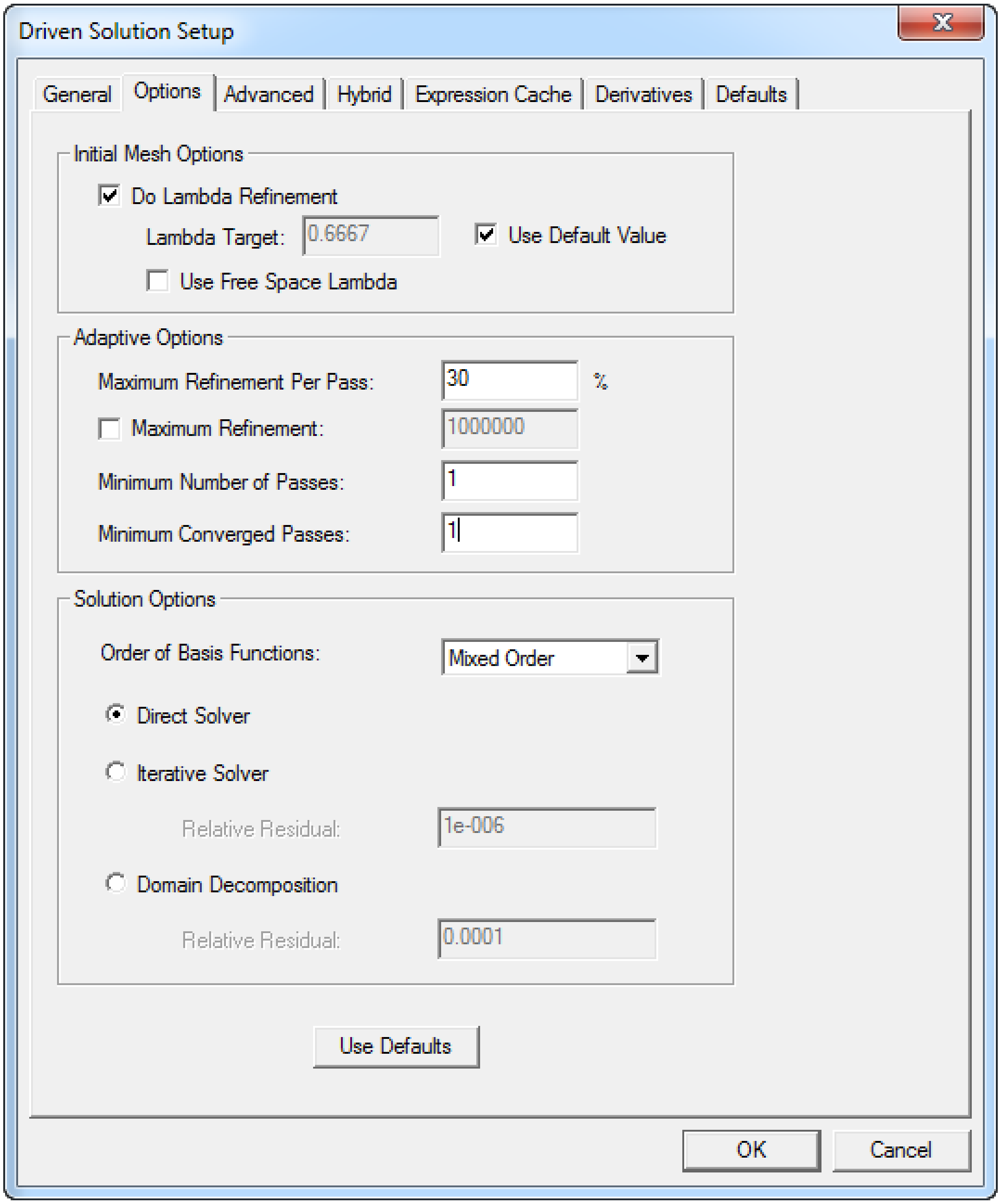


Figure 4-2 Options tab

Add a Frequency Sweep

1. Click **HFSS>Analysis Setup>Add Sweep**

   The **Edit Frequency Sweep** dialog box appears.

2. Enter the following fields in Edit Frequency Sweep dialog box as shown in the figure below.

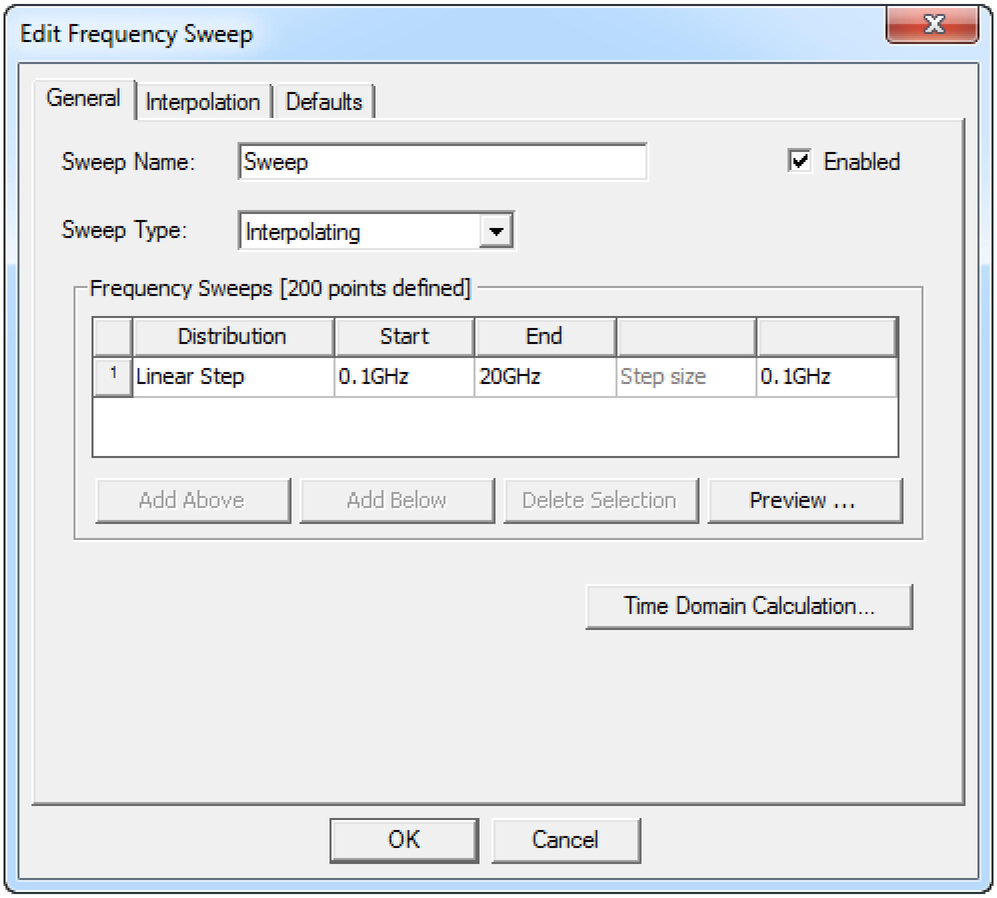


Figure 4-3 Edit Frequency Sweep

3. Click the **Interpolation**tab and edit the fields as in shown in the figure below and click **OK**.

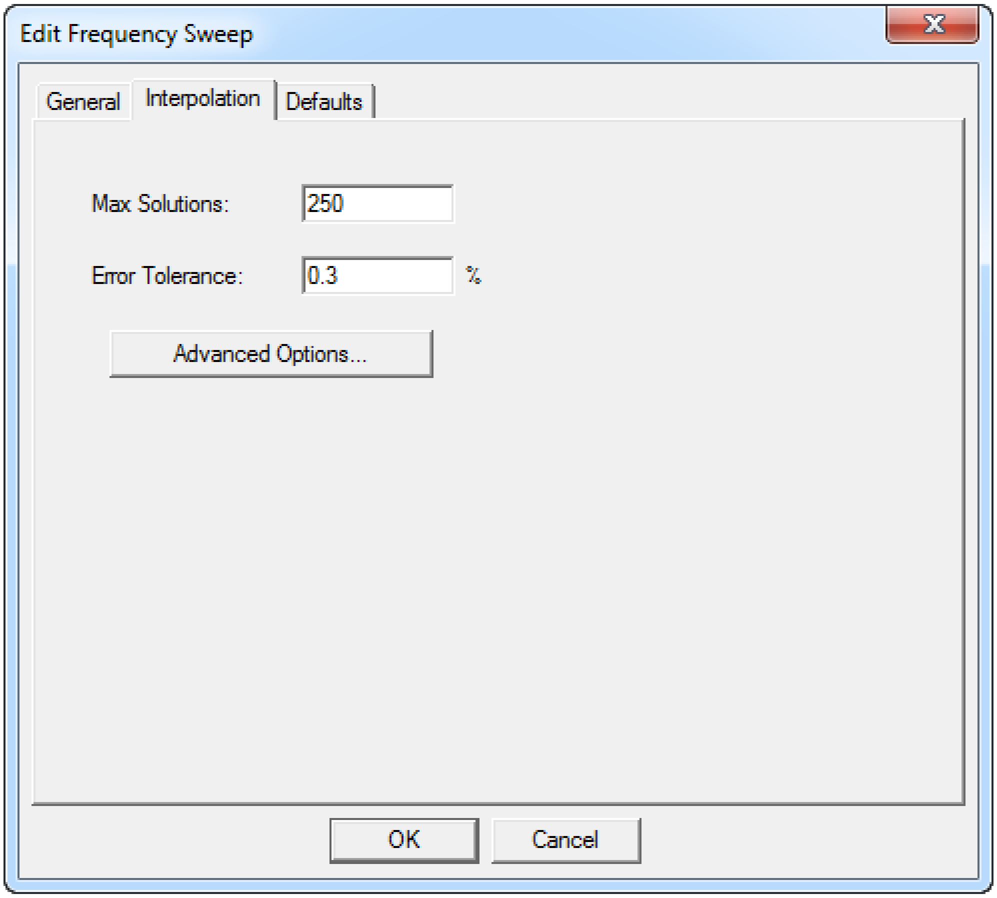


Figure 4-4 Interpolation

Model Validation

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

To validate the model:

1. Click **HFSS>Validation Check**

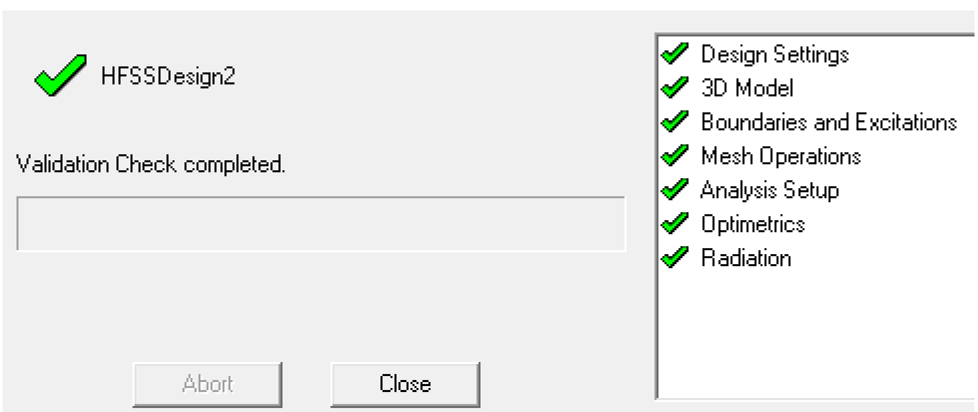


Figure 4-5 Validation Check

2. Verify whether your dialog box is the same as Figure 5.
3. Click **Close.**

Note: For this project, ignore warnings as no action is required.

Analyze the Model

To start the solution process:

1. Click **HFSS>Analyze All**

   Note Change the design name to *No_Solve_Inside(Driven Terminal)* and then, save the project *si_spiral_inductor*.

Review Solution Data

To view the Solution Data:

1. Click **HFSS>Results>Solution Data**
2. Click **Profile**, **Convergence**, **Matrix Data** etc to see those panels and the results they contain.

Review the Profile Panel

The Profile window lets you view a synopsis of the simulation results ranging from mesh creation and refinement to information about the different adaptive passes, the matrix assembly and solve along with extraction of electromagnetic field and SYZ parameter data. The more highly refined the mesh, i.e. higher the number of tetrahedra, more accurate is HFSS' solution of the design generating optimum results. However, there is a trade-off in the number of tetrahedra used and the computational resources required. Higher the number of tetrahedra the more accurate the solutions. Keep in mind that increased accuracy requires more computational resources and more time.

Task	Real Time	CPU Time	Memory	Information
				Solution Basis Order: Mixed
Mesh Refinement				Manual Seed Based
Mesh (volume, seed)	00:00:00	00:00:00	27.3 M	2882 tetrahedra
Mesh Refinement				Lambda Based
Mesh (lambda based)	00:00:01	00:00:01	29.2 M	4017 tetrahedra
Simulation Setup	00:00:00	00:00:00	32.9 M	Disk = 0 KBytes
Port Adaptation	00:00:00	00:00:00	42.8 M	Disk = 3 KBytes, 3551 tetrahedra
Mesh (port based)	00:00:02	00:00:02	28.8 M	4195 tetrahedra
				Length1
Adaptive Pass 1				Frequency: 12GHz
Adaptive Meshing Frequency: 12GHz on...				
Simulation Setup	00:00:00	00:00:00	33.6 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:01	64.6 M	Disk = 67 KBytes, 3710 tetrahedra , 2 lumped ports
Solver DCS8	00:00:00	00:00:01	96.3 M	Disk = 0 KBytes, matrix size 10752 , matrix bandwidth 16.8
Field Recovery	00:00:00	00:00:01	96.3 M	Disk = 2417 KBytes, 2 excitations , Average Order 0.280054
Adaptive Pass 2				Frequency: 12GHz
Mesh (volume, adaptive)	00:00:00	00:00:00	30.2 M	5314 tetrahedra
Adaptive Meshing Frequency: 12GHz on...				
Simulation Setup	00:00:00	00:00:00	35 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:01	90.1 M	Disk = 0 KBytes, 4783 tetrahedra , 2 lumped ports
Solver DCS8	00:00:00	00:00:03	142 M	Disk = 0 KBytes, matrix size 19497 , matrix bandwidth 18.5
Field Recovery	00:00:00	00:00:01	142 M	Disk = 2695 KBytes, 2 excitations , Average Order 0.448881
Adaptive Pass 3				Frequency: 12GHz
Mesh (volume, adaptive)	00:00:00	00:00:01	30.9 M	6040 tetrahedra
Adaptive Meshing Frequency: 12GHz on...				
Simulation Setup	00:00:00	00:00:00	36 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:01	110 M	Disk = 0 KBytes, 5469 tetrahedra , 2 lumped ports
Solver DCS8	00:00:00	00:00:04	192 M	Disk = 0 KBytes, matrix size 25658 , matrix bandwidth 19.4
Field Recovery	00:00:00	00:00:01	192 M	Disk = 3000 KBytes, 2 excitations , Average Order 0.532821
Adaptive Pass 4				Frequency: 12GHz
Mesh (volume, adaptive)	00:00:01	00:00:01	31.7 M	6921 tetrahedra
Adaptive Meshing Frequency: 12GHz on...				
Simulation Setup	00:00:00	00:00:00	37 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:01	128 M	Disk = 0 KBytes, 6295 tetrahedra , 2 lumped ports
Solver DCS8	00:00:00	00:00:05	265 M	Disk = 0 KBytes, matrix size 31898 , matrix bandwidth 19.9
Field Recovery	00:00:00	00:00:01	265 M	Disk = 3431 KBytes, 2 excitations , Average Order 0.596346
Adaptive Pass 5				Frequency: 12GHz

Figure 4-6 Profile

Review the Convergence Panel

To view the Convergence data click the **Convergence** tab.

Note: The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

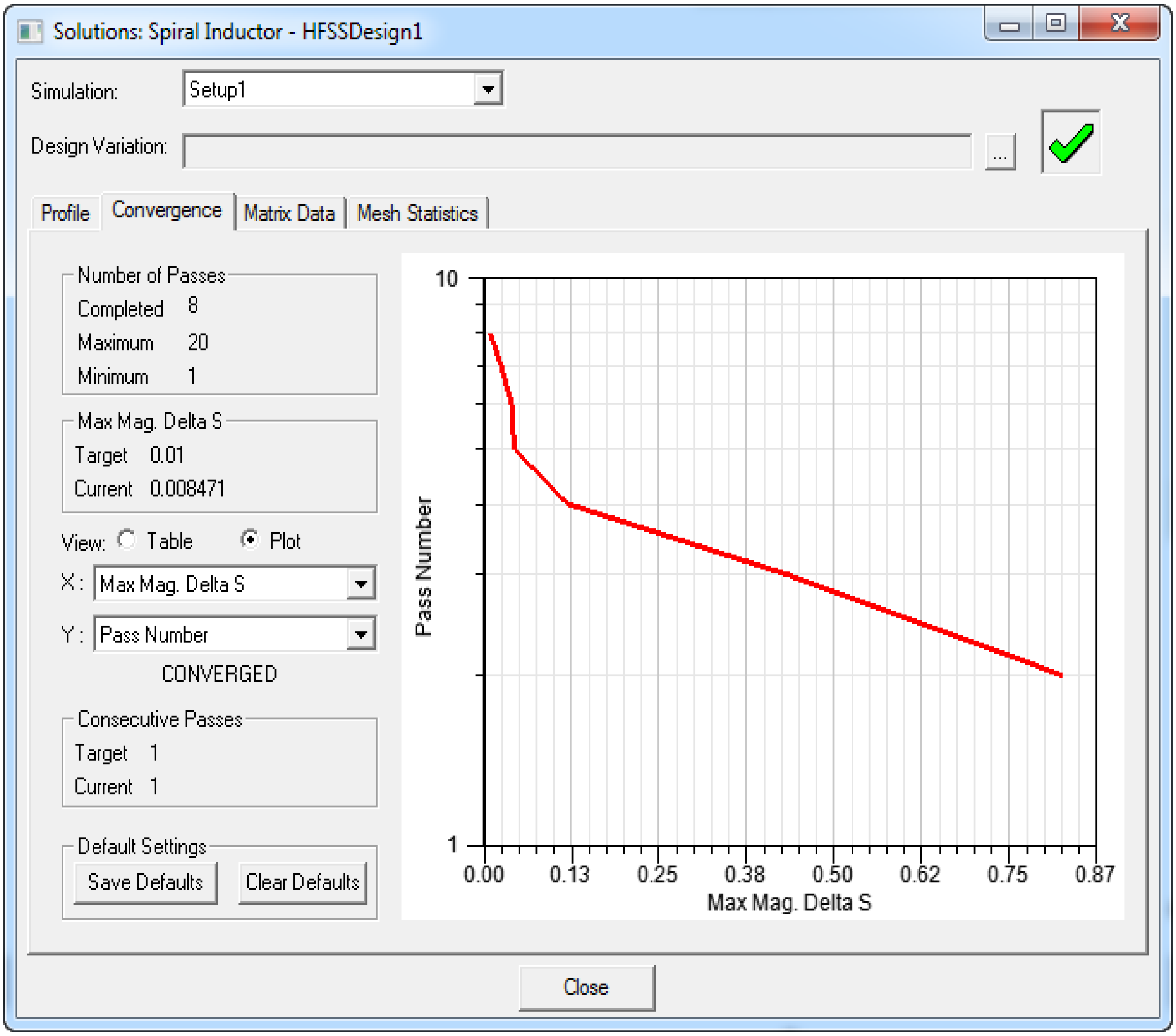


Figure 4-7 Convergence dialog box

Review the Matrix Data Panel

To view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution, click the **Matrix Data** tab.

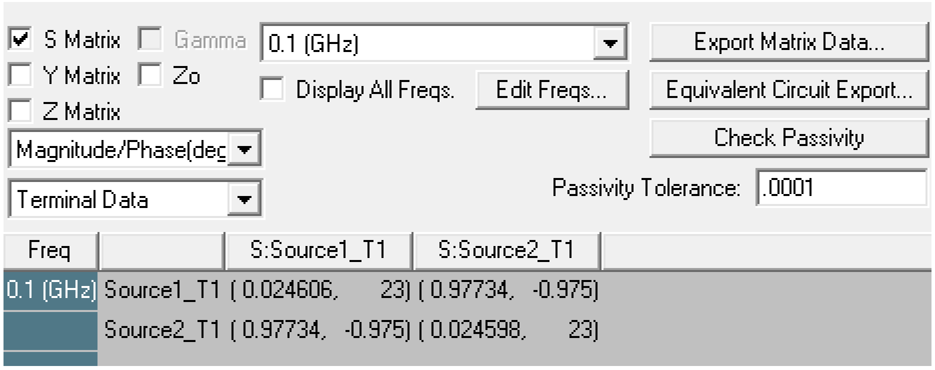
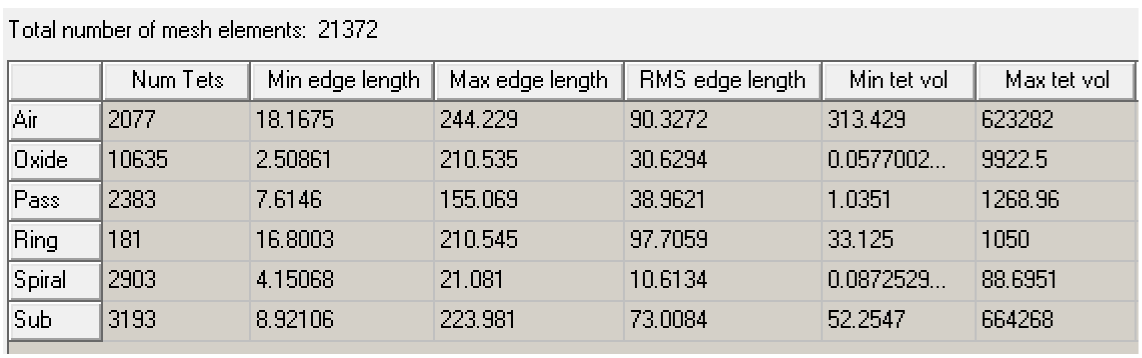


Figure 4-8 Matrix Data

Note: To view a real-time update of the Matrix Data, set the Simulation to **Setup1, Last Adaptive.**

Review the Mesh Statistics Panel

As the title indicates this panel shows statistics of the mesh, more specifically, it gives break-ups of the tetrahedra used to solve the different components of the model and their size and data.



Total number of mesh elements: 21372

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol
Air	2077	18.1675	244.229	90.3272	313.429	623282
Oxide	10635	2.50861	210.535	30.6294	0.0577002...	9922.5
Pass	2383	7.6146	155.069	38.9621	1.0351	1268.96
Ring	181	16.8003	210.545	97.7059	33.125	1050
Spiral	2903	4.15068	21.081	10.6134	0.0872529...	88.6951
Sub	3193	8.92106	223.981	73.0084	52.2547	664268

Figure 4-9 Mesh Statistics

Generate Reports

The subsequent sections describe how to create different reports, customize the equations for the Y axis and create output variables.

Create S-parameter vs. Frequency Plot

1. Click **HFSS>Results>Create Terminal Solution Data Report>Rectangular Plot**

   The Report dialog box appears.

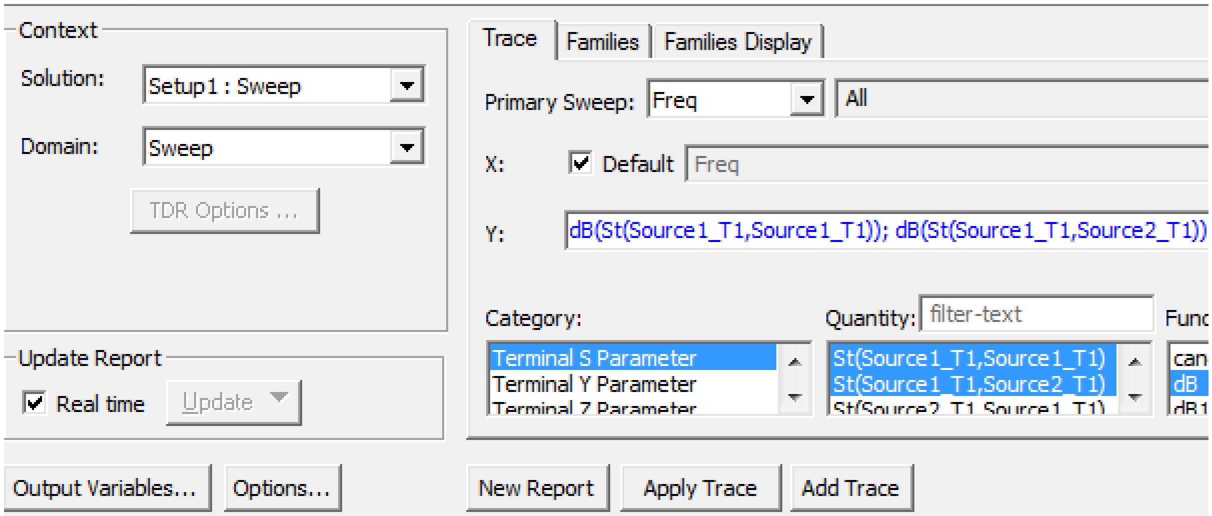


Figure 4-10 Report dialog box

2. Edit the fields as shown in "Report dialog box" above. .
3. Click **New Report and** Click **Close**.

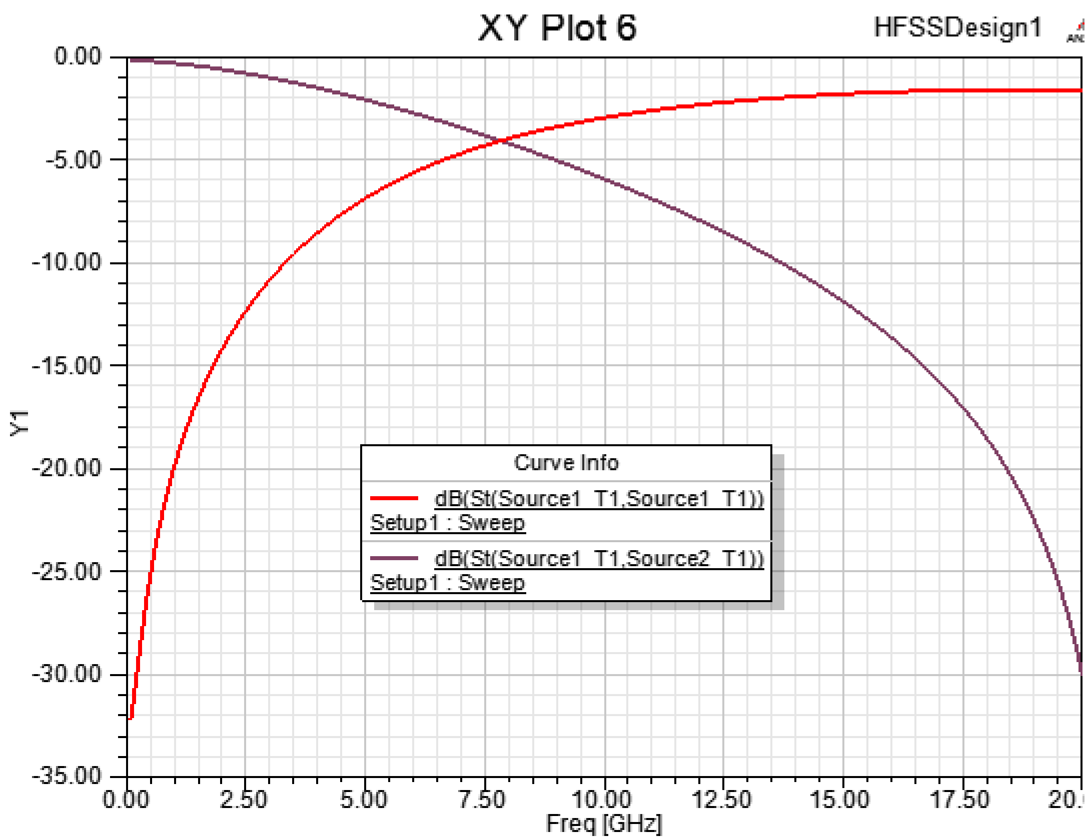


Figure 4-11 The XY Plot

Custom Equations – Output Variables

1. Click **HFSS>Results>Create Terminal Solution Data Report>Rectangular Plot**
   The **New Report** dialog box appears.
2. Click **Output Variables**.

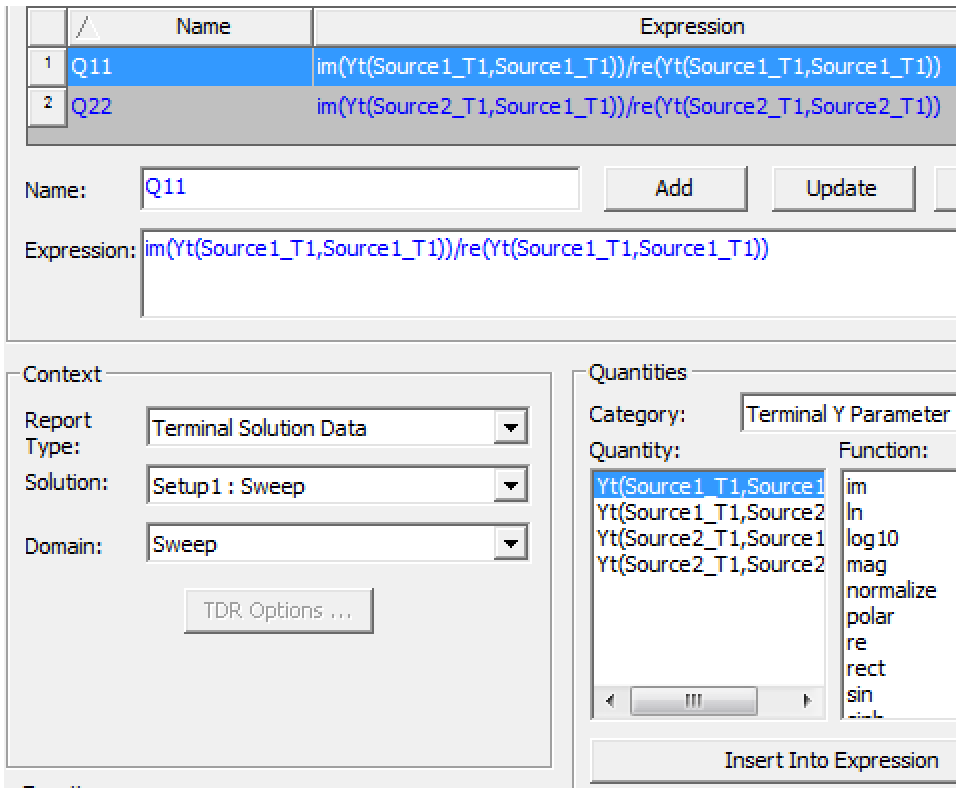


Figure 4-12 Output Variables dialog box

3. Enter *Q11* in the **Name** field.
4. Select *Terminal Y Parameters* from **Category**.
5. Select *Yt(Source1_T1, Source1_T1)* as **Quantity.**
6. Select *im* from the **Function** list.
7. Click **Insert Quantity into Expression**.
8. Type the forward slash(**/**).

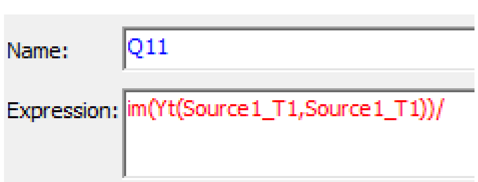


Figure 4-13 Expression

Note Notice the expression is in red ink because it is incomplete. The slash causes HFSS to expect another function. Red ink indicates inaccuracy or incompleteness.

9. Select *Yt(Source1_T1, Source1_T1)* in the **Quantity** field.
10. Select **Function:***re*
11. Click **Insert Quantity into Expression**.
12. Click **Add**.

    The output variable **Q11** is added to the list.
13. Create **Q22** with *Yt(Source2_T1, Source2_T1)* as quantity.
14. Click **Add.**

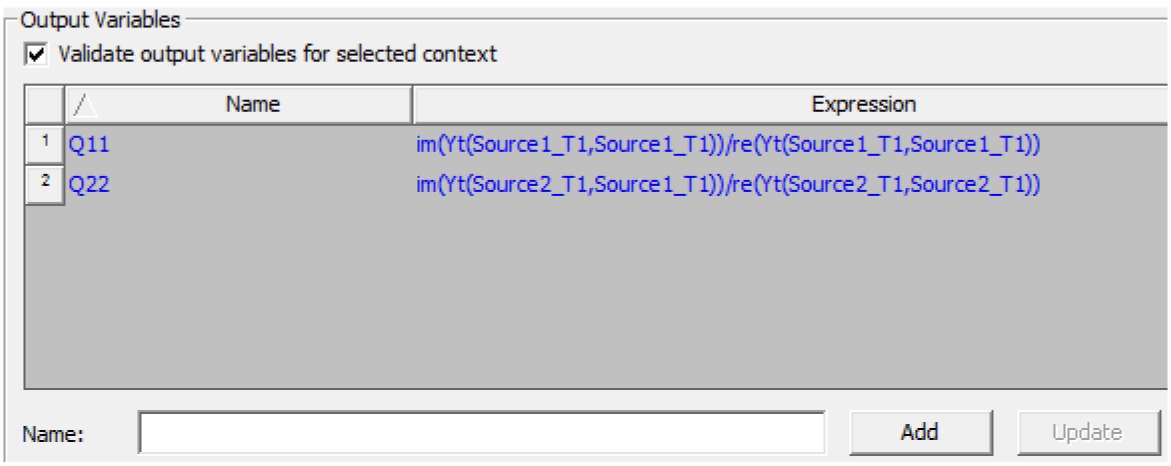


Figure 4-14 Output Variables set

15. Click **Done.**

    The **Output Variables** dialog box closes.
16. Edit the fields in the **Report** dialog box as in "Report dialog box" on the facing page. .
17. Click **New Reports** and click **Close**.

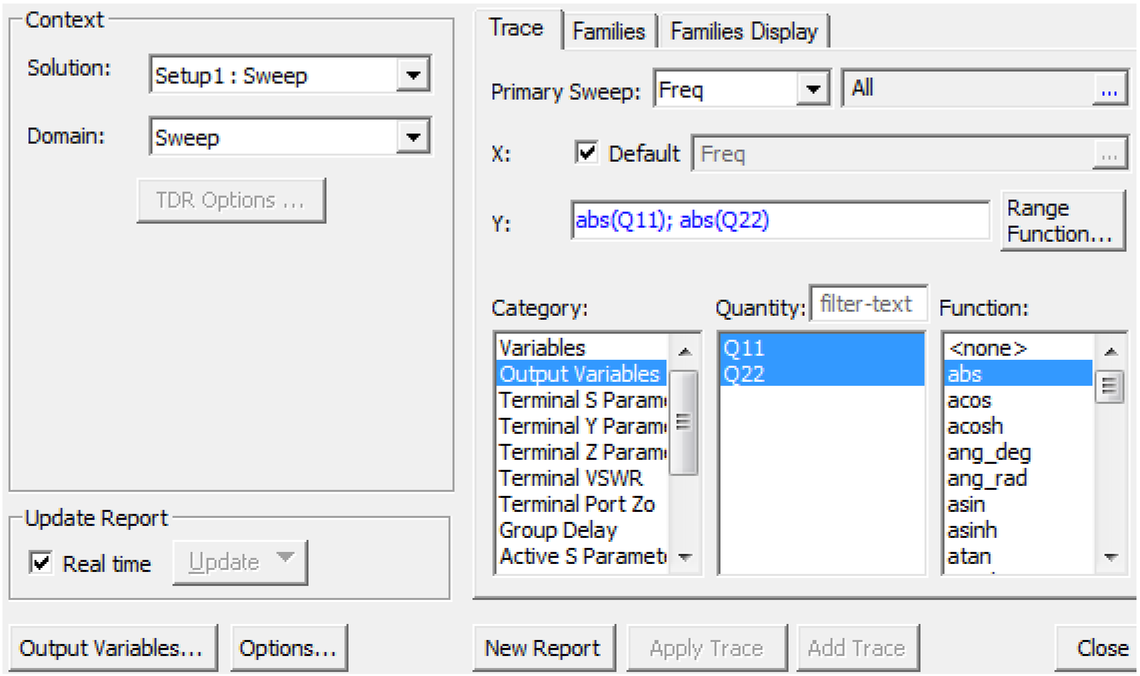


Figure 4-15 Report dialog box

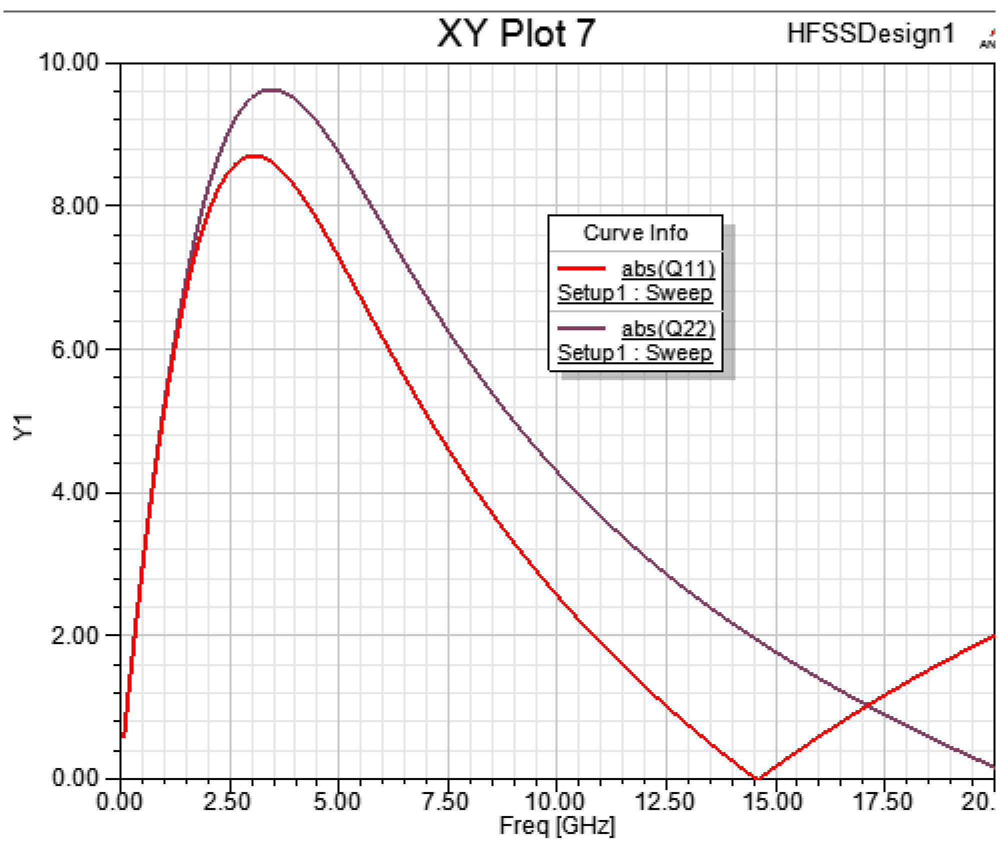


Figure 4-16 Y parameters versus Frequency

Use Output Variables for Next Report

To use Output Variables for another report:

1. Click **HFSS>Results>Create Terminal Solution Data Report>Rectangular Plot**
2. In the **New Report** window**, Trace Tab** click the **Output Variables** button
3. In the **Output Variables** dialog box enter these values:
   - **Name:***L11*
   - Type **-1/(2*pi*freq*** in the **Expression** field**.**
   - Select *Terminal Y Parameters*from **Category.**
   - Select *Yt(Source1_T1, Source1_T1)* from **Quantity**.
   - Click*im* from the **Function** list.
4. Click **Insert into Expression**.
5. Type a bracket “)” key and click **Add.**

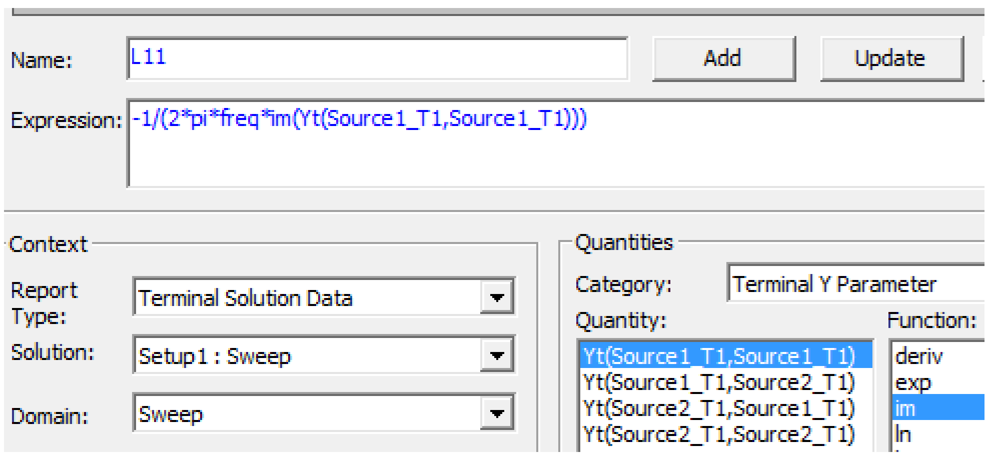


Figure 4-17 New Report dialog box

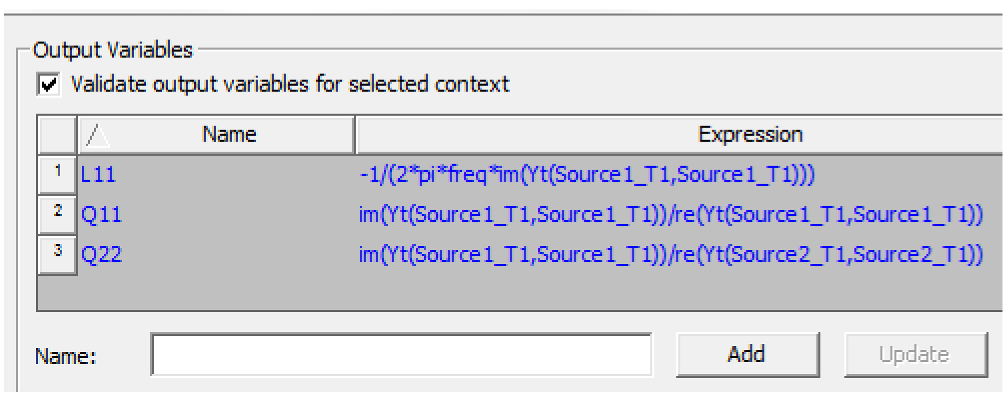


Figure 4-18 Output Variables dialog box

6. Click **Done** to close the **Output Variables** dialog box.
7. Edit the fields in **Report** dialog box as shown in the following figure.

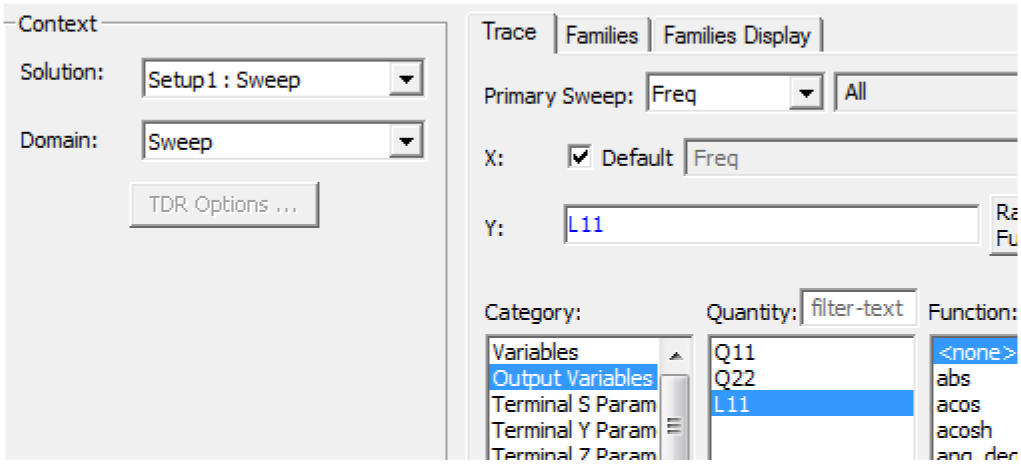


Figure 4-19 Report dialog box

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

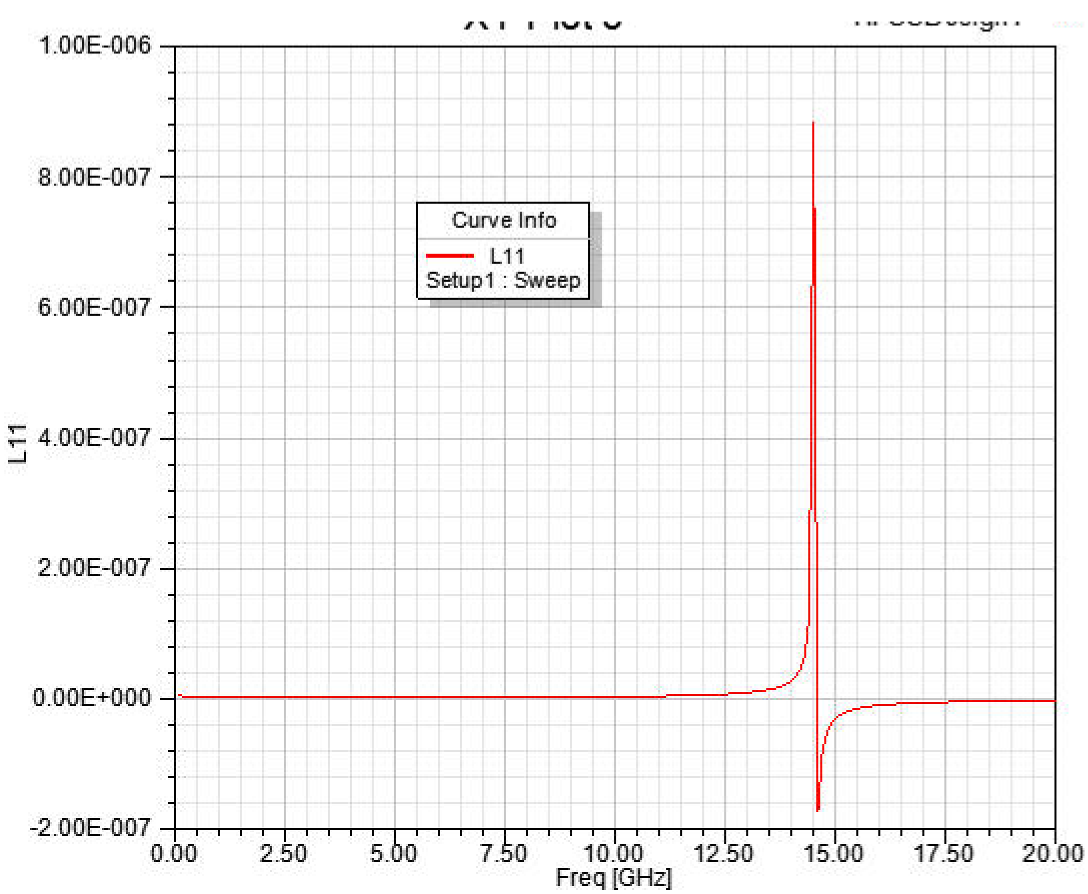
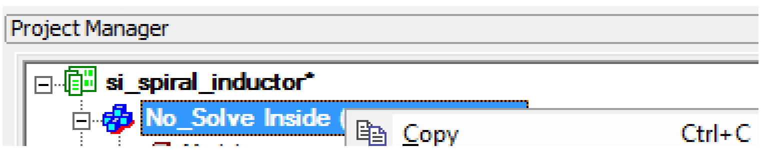


Figure 4-20 L11 versus Frequency

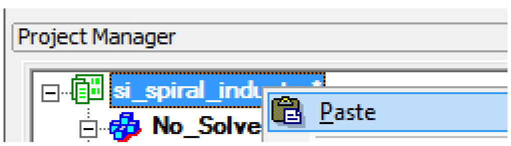
Simulate with Solve Inside Conductors

In this section, we will select simulate the design with **Solve Inside** selected for the spiral. By default **Solve Inside** gets unchecked for metals or highly conductive materials. In such cases, 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 for **Solve Inside**. When **Solve Inside** is selected it includes tetrahedra inside a conductor for simulation which may require a large mesh. **Solve Inside** can be useful for low frequency analysis of electrically small projects for enhanced accuracy of sensitivity design parameters such as the Q factor.

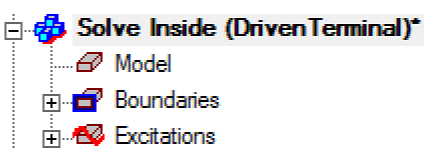
1. In the **Project Manager** window select the design and copy it.



2. Go the project folder and paste the design.



3. Rename the pasted design.

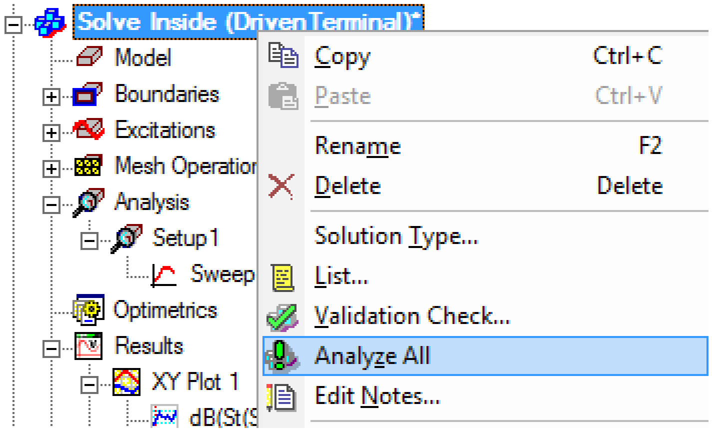


4. Double-click **spiral** from the history tree and select **Solve Inside** in the **Attribute** dialog box.

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

   Note For this project, ignore this message.

5. Right click the design and select **Analyze All** from the short-cut menu.



Results with Solve Inside

All the plots get updated real time as the simulation takes place. For this design it may take more passes to converge than when **Solve Inside** was unchecked.

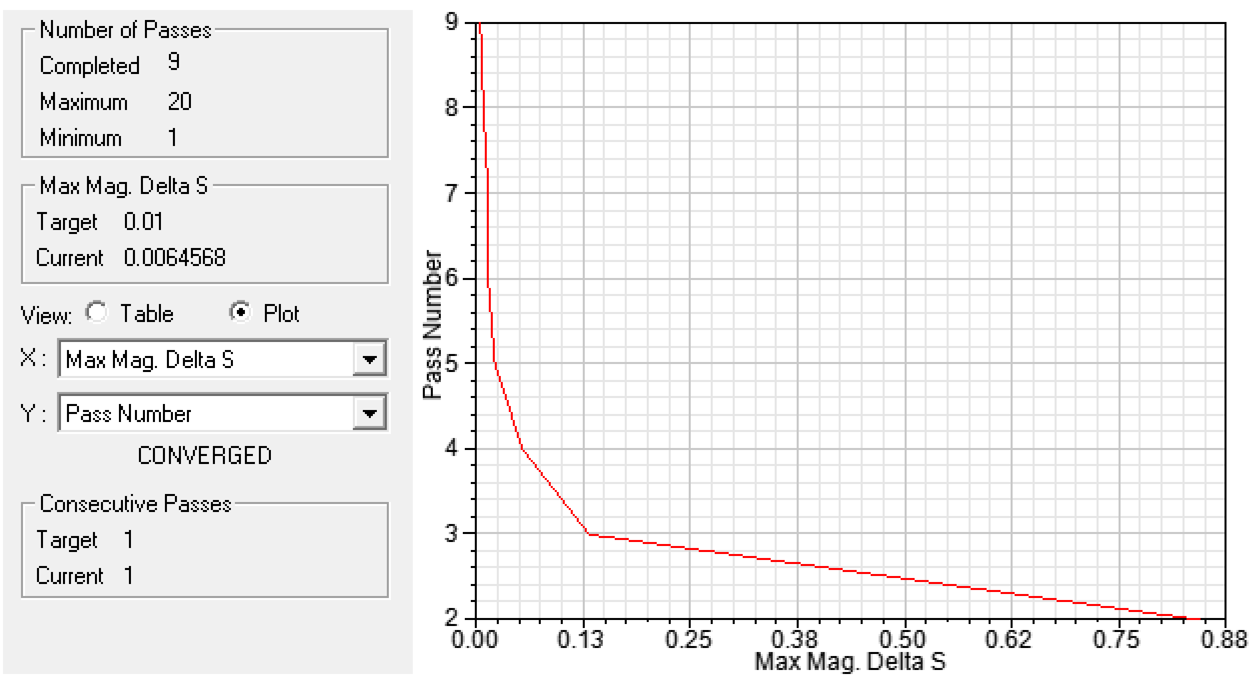
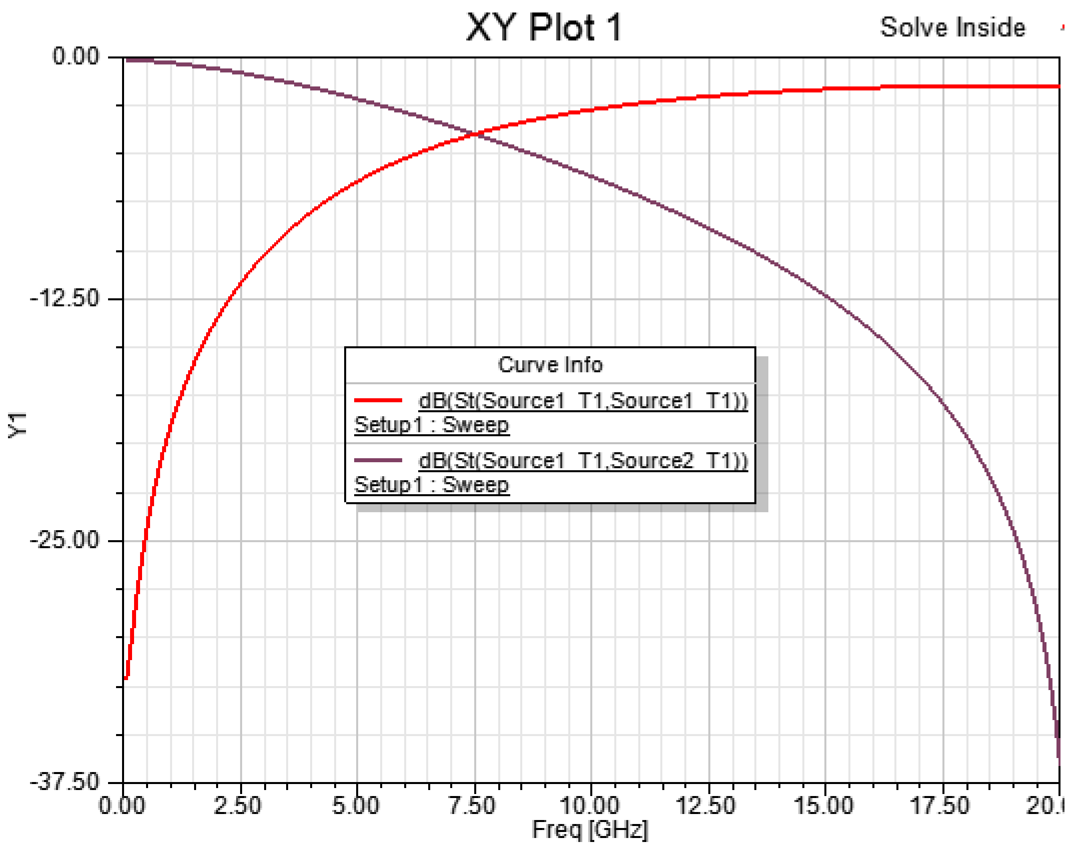
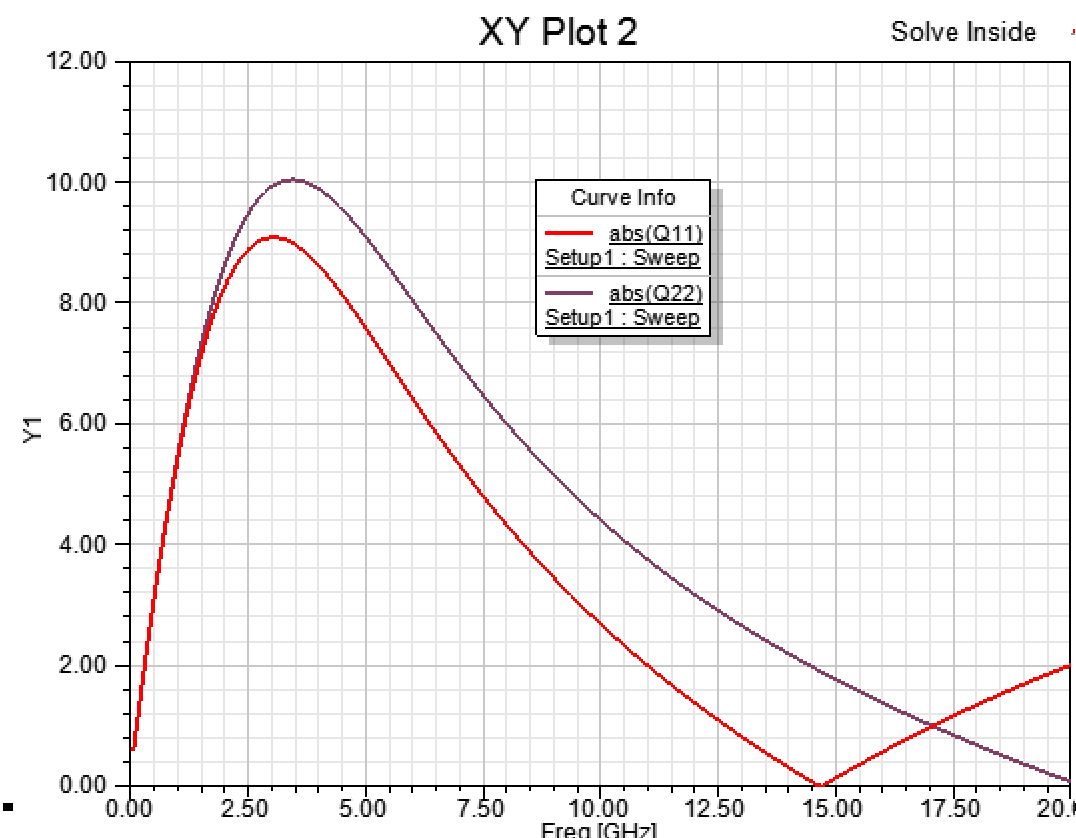


Figure 4-21 Convergence Plot


XY Plot 1
Solve Inside
0.00
-12.50
-25.00
-37.50
Y1
Curve Info
dB(St(Source1_T1,Source1_T1))
Setup1 : Sweep
dB(St(Source1_T1,Source2_T1))
Setup1 : Sweep
0.00
2.50
5.00
7.50
10.00
12.50
15.00
17.50
Freq [GHz]


XY Plot 2
Solve Inside
12.00
10.00
8.00
6.00
4.00
2.00
0.00
Y1
Curve Info
abs(Q11)
Setup1 : Sweep
abs(Q22)
Setup1 : Sweep
0.00
2.50
5.00
7.50
10.00
12.50
15.00
17.50
Freq [GHz]

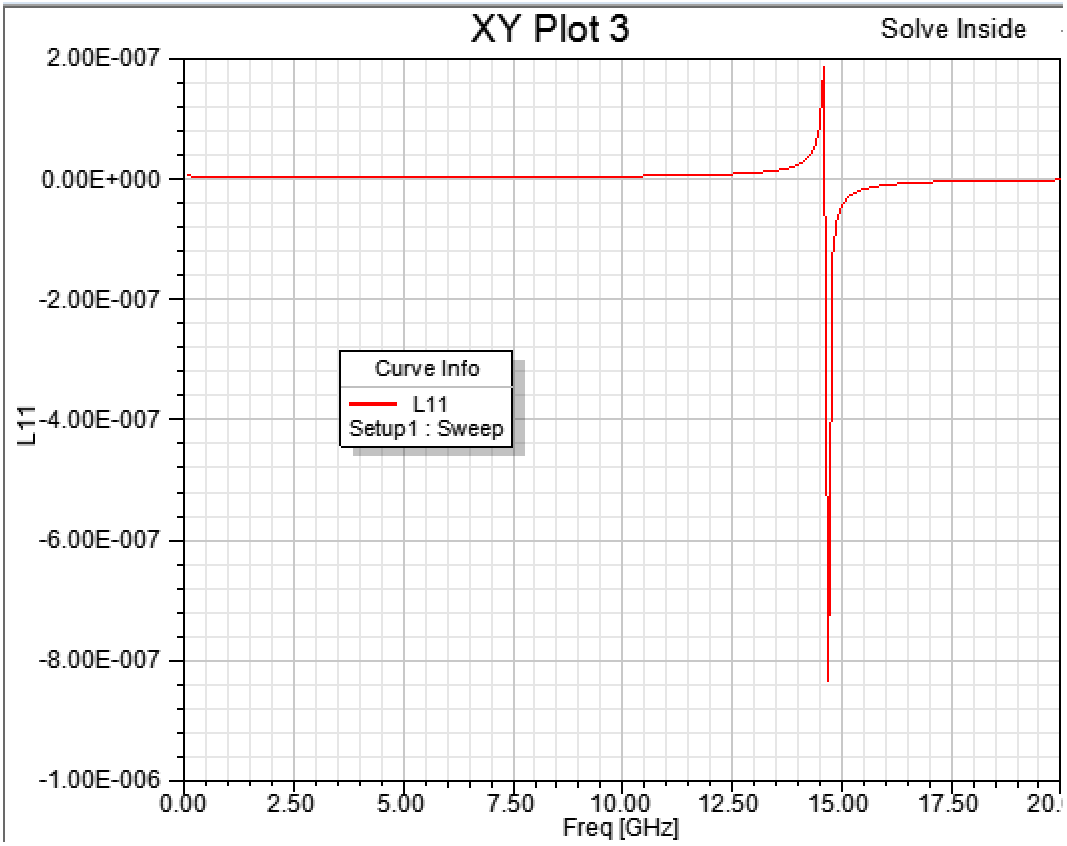


Figure 4-22 Plot with Solve Inside