

Getting Started with HFSS: Ultra High Frequency Probe

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 18.1
April 2017

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2008 companies.

Copyright and Trademark Information

© 2017 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS 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 in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

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. is certified to ISO 9001:2008.

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 18.1 - © 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 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.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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**.

ANSYS Electromagnetics Suite 18.1 - © 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: UHF Probe 1-1

2 - Set Up the Project **2-1**

Launch Ansys Electronics Desktop 2-1

Set Tool Options 2-2

Open and Save a New Project 2-3

Set the Model Units 2-4

Set Solution Type 2-4

3 - Create the 3D Model **3-1**

Creating Annular Rings 3-1

Create the First Cylinder (Ring_Inner) 3-1

Create the Second Cylinder (Ring 1) 3-3

Subtract Second Cylinder from First 3-4

Create Ring 2 3-6

Create Arm _1 3-9

Group the Conductors 3-10

Create Arm_2 3-12

Create the Grounding Pin 3-13

Group Conductors 3-15

Create a Circle 3-16

Draw the Enclosure for Air 3-17

Create Radiation Boundary 3-19

Assign Excitation 3-21

Rename the Wave Port and Terminal 3-24

Create Infinite Ground Plane 3-25

Create a Radiation Setup 3-28

Add Length Based Mesh Operation to the Radiation Boundary 3-29

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Analyze the Model **4-1**
Create an Analysis Setup 4-1
Add Frequency Sweep 4-1
Validate Model 4-2
Analyze the UHF Probe 4-3
View Solution Data 4-3
Create Reports 4-6
Terminal S Parameter Vs Frequency 4-6
Change the X-axis Label Settings 4-7
Create Far Field Overlay 4-9

ANSYS Electromagnetics Suite 18.1 - © 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 ultra high frequency probe.

Sample Project: UHF Probe

An ultra high frequency probe is a dipole antenna that operates in the ultra-high frequency range.

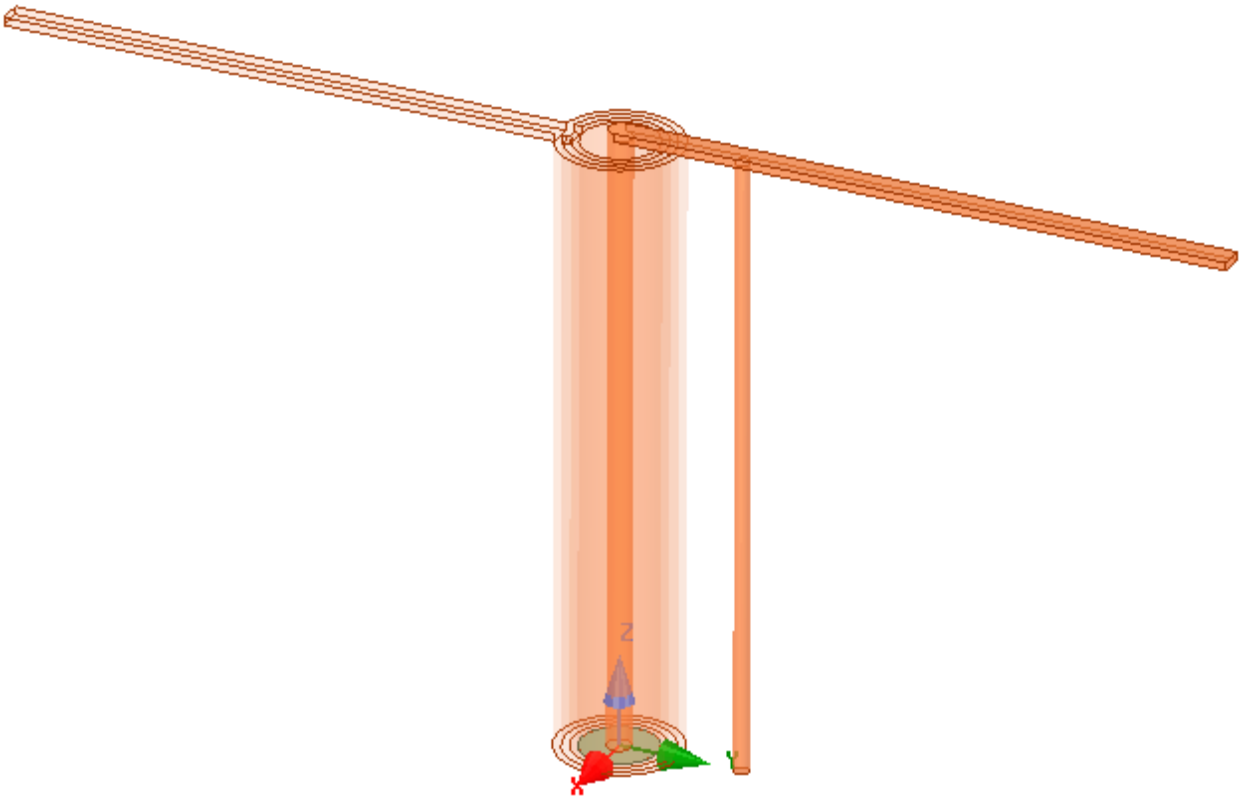


Figure 1-1 UHF Probe Model

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 18.1 - © 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 HFSS and Set up the Project
- Launch HFSS
- Set Tool Options
- Open and Save a New Project
- Set the Model Units

Launch HFSS and Set up the Project

Launch HFSS and insert a project to create a design of the ultra high frequency probe. To set up the project perform the following tasks:

- Launch HFSS
- Set Tool Options
- Open and Save a New Project
- Set Modeler units
- Select Solution Type

The following sections show you how to perform the above tasks.

Launch Ansys Electronics Desktop

We recommend you store a shortcut of the Ansys Electronics Desktop application on your desktop.

1. Double-click the **Ansys Electronics Desktop** icon on your desktop.
2. Click Projectn and press F2 to rename the project as you wish.

   Electronics Desktop with the renamed project is shown in the figure below.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

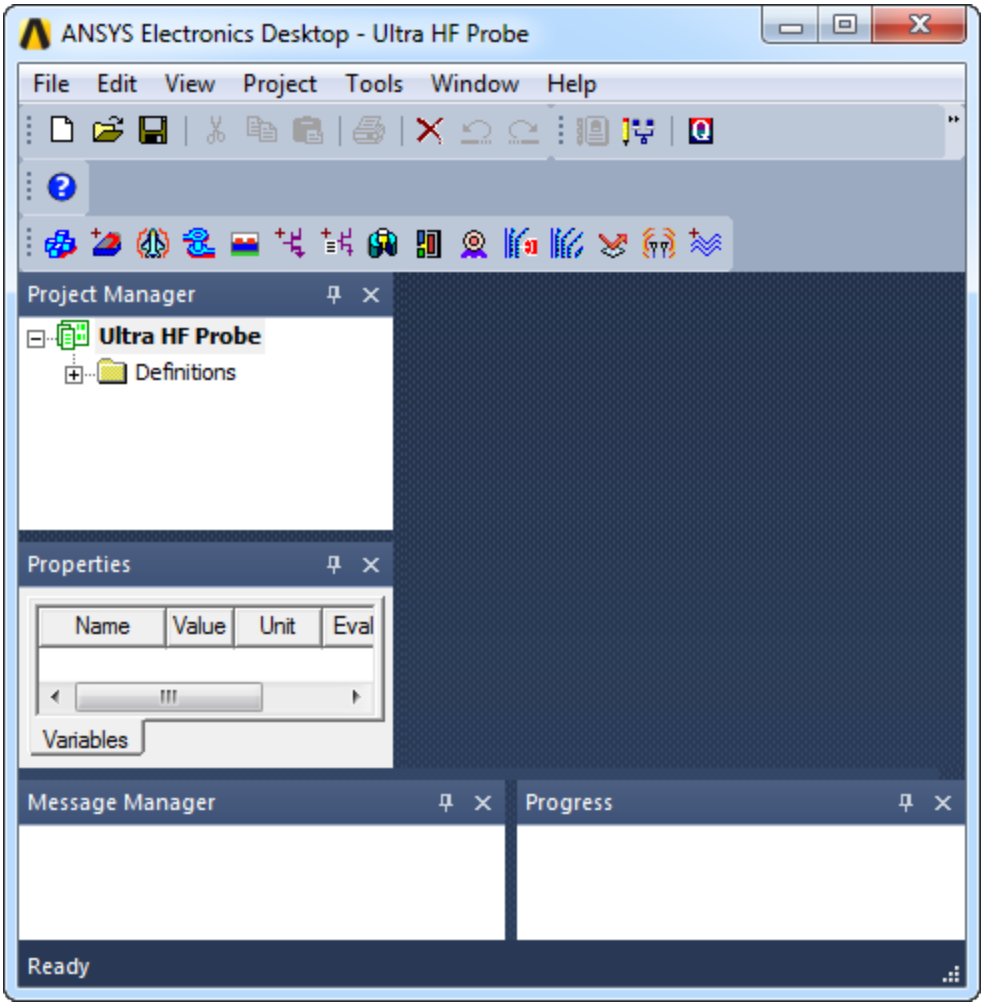


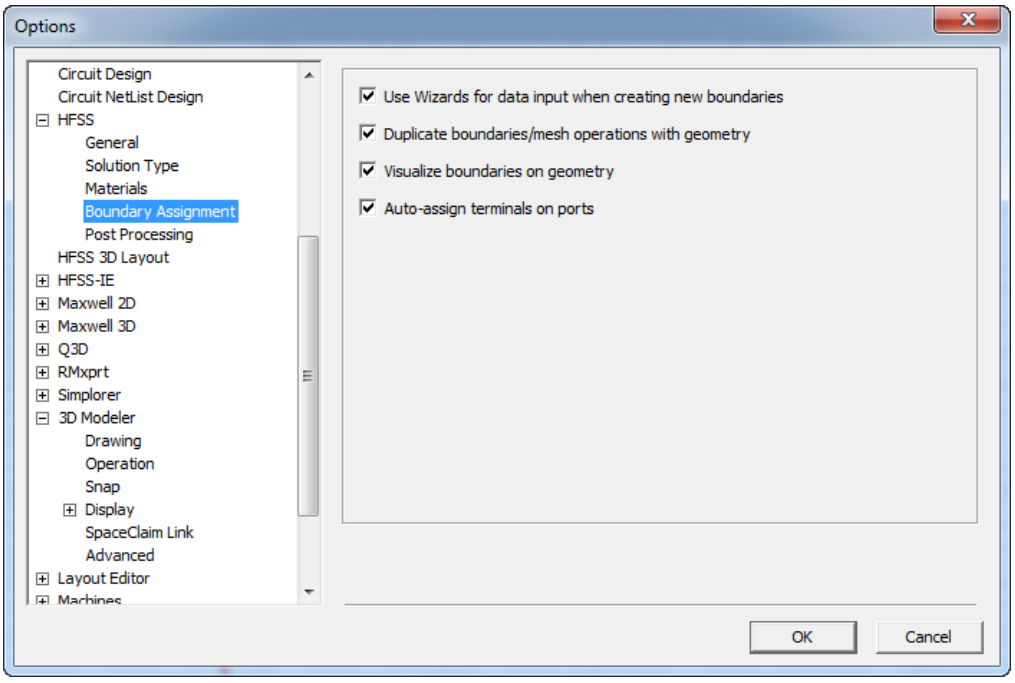
Figure 2-1 Project Manager Window

Set Tool Options

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

1. Click **Tools>Options>General Options.**
   The **Options** window appears.
2. Expand **HFSS** and click **Boundary Assignment** .
3. Ensure all boundary assignment options are selected.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Expand **3D Modeler Options** and select **Drawing**.
5. Ensure **Automatically cover closed Polyline** and **Edit property of new primitives** check boxes are selected.

   Note The **Edit property** option causes a **Properties** dialog box to appear when you create a new object in the Modeler.

6. Click **OK**.

Open and Save a New Project

If a project folder does not appear in a Project tree when you launch Electronics Desktop, you can go to **File** and click **New** to include one; if the **Project Manager** window does not appear, go to **View** and enable it.

Now include an HFSS design type for the project as follows.

1. Go to the menu item Project and select **Insert HFSS Design**.

   HFSSDesign*n* appears in the **Projecttree**. See "HFSS Designn included." on the next page. .
2. Go to **View > Coordinate System** to select the size of the CS.
3. Similarly, go to View > Grid Settings to adjust grid visibility.
4. Click **File>Save As**.
   The **Save As** dialog box appears.
5. Browse for a location where you can save the file.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

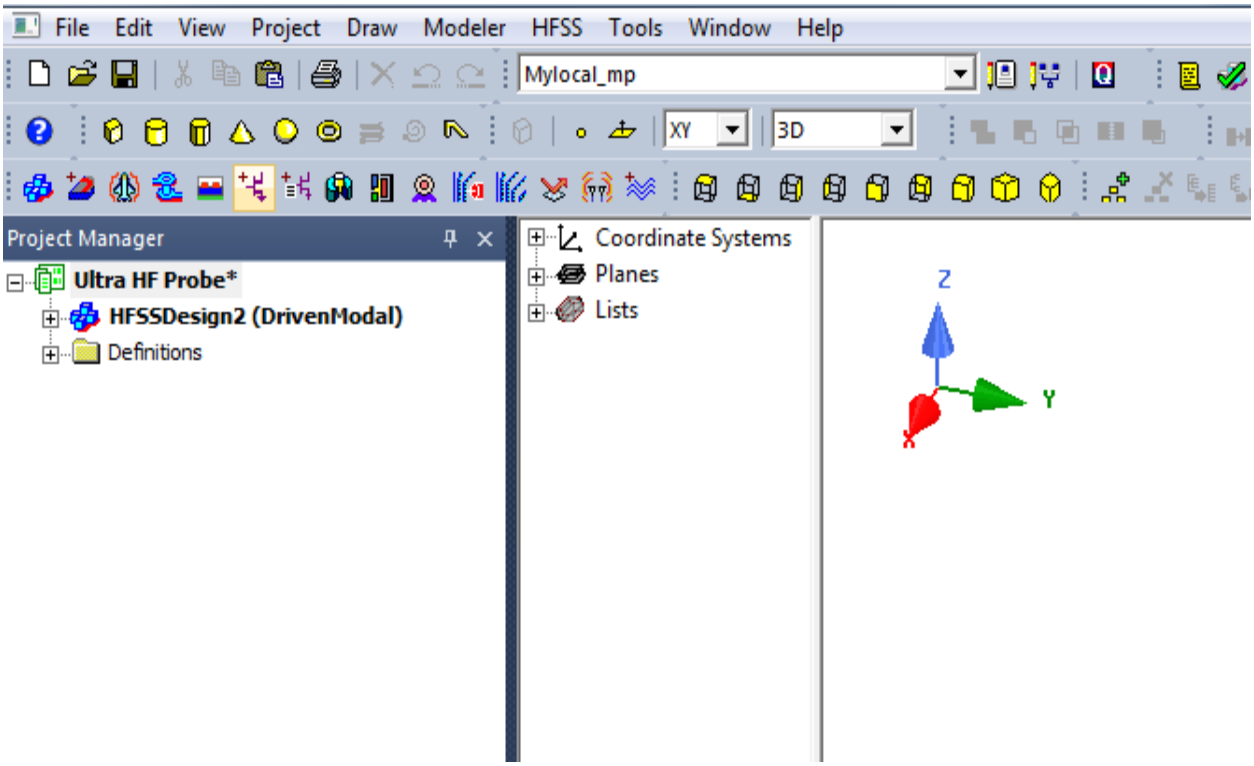


Figure 2-2 HFSS Design*n* included.

Set the Model Units

Define the units for the geometric model as follows:

1. On the HFSS toolbar, click **Modeler>Units**.
   The **Set Model Units** dialog box appears.



Figure 2-3 Set Model Units dialog box

2. Select **in** (inches) from the **Select units** drop-down menu, and click **OK**.

Set Solution Type

To set the solution type:

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **HFSS>Solution Type** on the toolbar.

   The **Solution Type** dialog box appears.

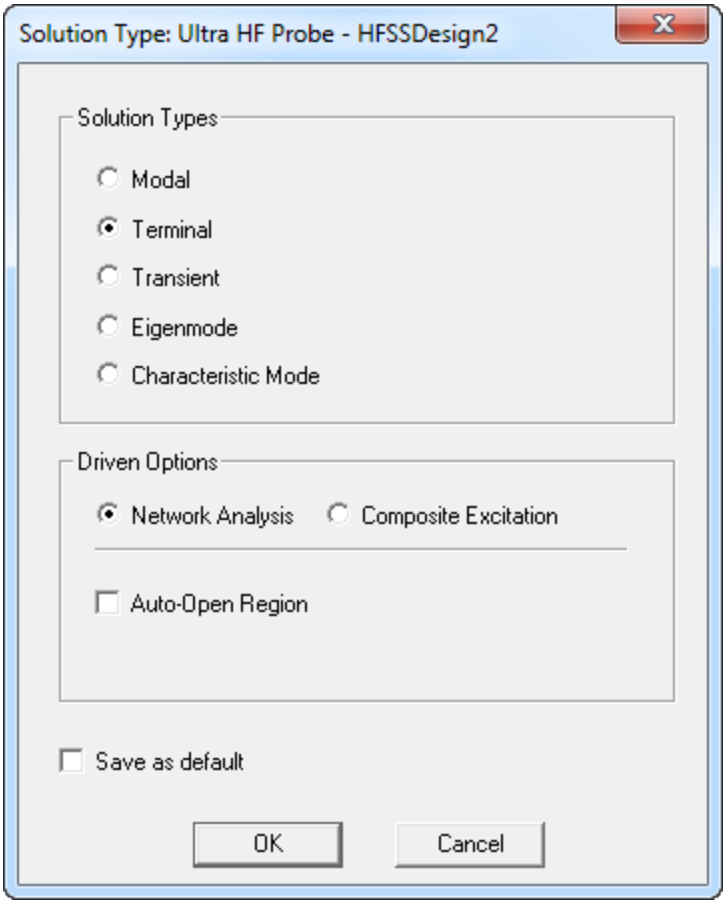


Figure 2-4 Solution Type dialog box

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

   Note Terminal solutions calculate terminal-based S-parameters of multi-conductor transmission line ports. The S-matrix solutions are expressed in terms of terminal voltages and currents.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Create the 3D Model

This chapter contains the following topics:

- Create the 3D Model
- Create Annular Rings
- Create the First Cylinder(Ring_Inner)
- Create the Second Cylinder
- Subtract Second Cylinder from First
- Create Ring2
- Create Arm_1
- Group the Conductors
- Create the Center Pin
- Create Arm_2
- Create Grounding Pin
- Group the Conductors
- Create a Circle
- Draw the Enclosure for Ait
- Create Radiation Boundary
- Assign Excitation
- Rename the Wave Port and Terminal
- Create Infinite Ground Plane
- Create a Radiation Setup
- Add Length Based Mesh Operation to Radiation Boundary

Create the 3D Model

You can now create the 3D model geometry. The following sections describe the steps of creating the geometry.

Creating Annular Rings

First create a cylinder to represent an outer radius of the probe and another cylinder for the inner radius. Then, perform a Boolean subtraction to generate an annular ring.

For this model, two sets of rings are necessary. Instead of manually creating both rings, create a ring, copy it, and edit the dimensions of the copy.

Create the First Cylinder (Ring_Inner)

1. Click **Draw>Cylinder**.

   The cursor contains a dotted line to draw the cylinder.

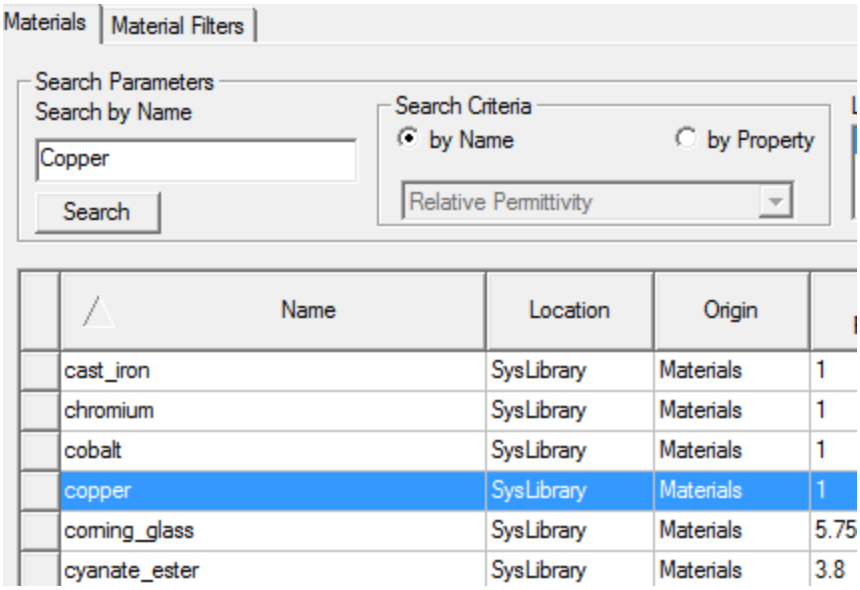
ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click inside the **3-D Modeler** window.
   The **YZ** mini axes is established.
3. Drag the cursor to draw the circle, and click the mouse.
   The mini **X** axis is established for the height.
4. Drag the cursor to complete the cylinder and, click the mouse.
   The **Properties** dialog box appears.
5. Click the **Command**tab.
6. Edit the fields in the **Command** dialog box as in the following figure.

	Name	Value	Unit	Evaluated V...
	Command	CreateCylinder		
	Coordinate System	Global		
	Center Position	0 ,0 ,0	in	0in , 0in , 0in
	Axis	Z		
	Radius	0.31	in	0.31in
	Height	5	in	5in
	Number of Segments	0		0

Figure 3-1 Command tab for the First Cylinder

7. Click **Attribute** and enter *ring_inner* in the **Name** field.
8. Select **Edit** from the **Materials** drop-down menu.
   The **Select Definitions** window appears.
9. Click the **Materials** tab.



ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3-2 Select Definitions

10. Type *Copper* in the **Search by Name** field.

    This highlights *copper*.

11. Click **OK** to close the **Select Definitions** dialog box.

    Copper is listed as the default material.

12. Click **OK** to close the **Properties** dialog box.

    The cylinder in the **Modeler** window is assigned copper as its material.

    Note In the **Message Manager** window the following message appears - *Solve inside for object 'ring_inner' is unset, due to material asssignment change*. This is because in the **Attributes** dialog box, the **Solve Inside** option gets unchecked by default when the material assignment is a metal. For this project, just ignore this message.

13. Click **View>Fit All>Active View** or do **CTRL+D**.

Create the Second Cylinder (Ring 1)

You can make a copy of the first cylinder and edit the copy for the outer ring.

1. Right-click the first cylinder and go to **Edit>Copy**.
2. Right-click anywhere in the modeler window and do **Edit>Paste**.

   This operation creates another identical cylinder in the same location.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	0 ,0 ,0	in	0in , 0in , 0in
	Axis	Z		
	Radius	0.37	in	0.37in
	Height	5	in	5in
	Number of Seg...	0		0

Figure 3-3 Command tab for Second Cylinder

3. Click the **CreateCylinder** command under **ring_inner1** from the history tree and change the radius to 0.37 in the **Properties** panel.

   Note: The **Properties** panel is docked under the **Project Manager** window.

4. Click **ring_inner1** and change the name to *ring_1* in the docked **Properties** window.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

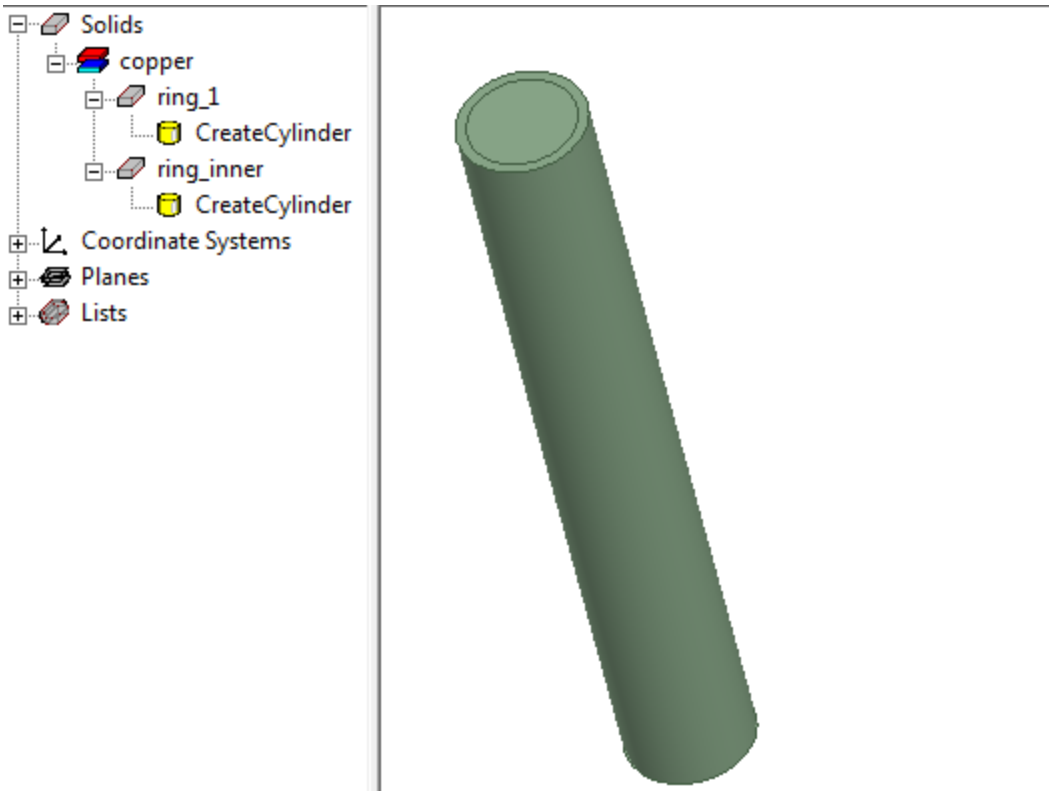


Figure 3-4 Structure with inner and outer cylinders

Subtract Second Cylinder from First

1. Click **Edit>Select>By Name**
   The **Select Object** dialog box appears.

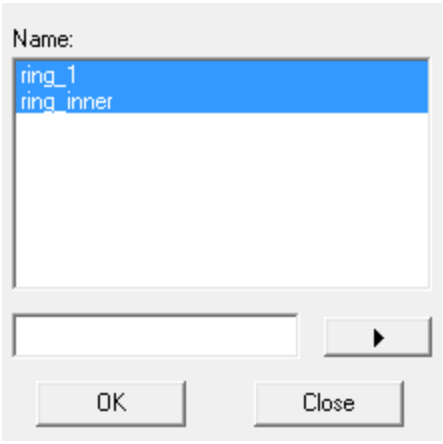


Figure 3-5 Select Object dialog box

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select *ring_1, ring_inner.*
3. Click **OK**.
4. Click **Modeler>Boolean>Subtract**
   The **Subtract** dialog box appears.
5. Set the **Subtract** dialog box as shown in the following figure.

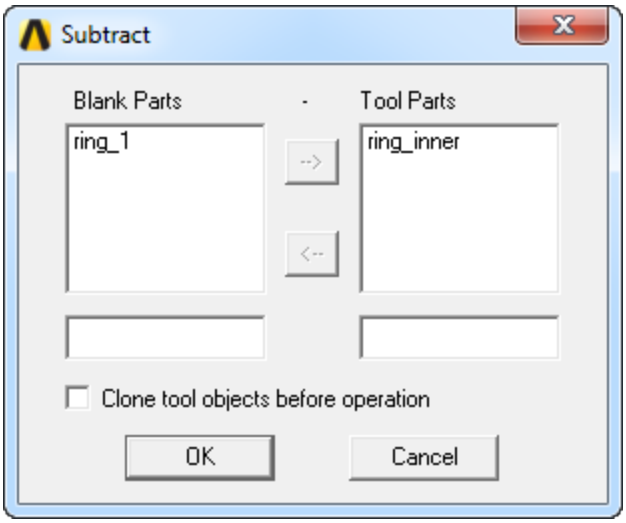


Figure 3-6 Subtract dialog box

6. Click **OK**.
   The subtract operation creates the annular ring.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

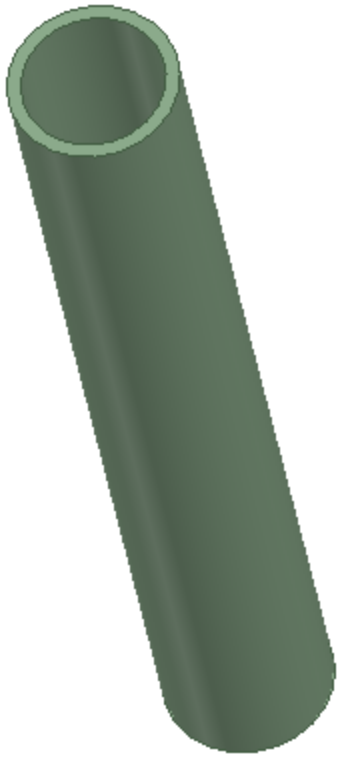


Figure 3-7 Annular Ring

Create Ring 2

Now create Ring2 in the same location as ring_1 as follows:

1. On the history tree, click *ring_1* and press **Ctrl+C**.
2. Press Ctrl+V to paste.

   The ring_2 gets pasted in the same location as ring_1 in the 3D Modeler, and the **History tree** lists ring_2.



Figure 3-8 History Tree

Note Be aware that after the copy-paste, *ring_1* becomes *ring_2* and the inner ring of *ring_2* becomes *ring_inner1*.

4. Expand the model tree as shown below.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note The order of editing is important. If you make the inner radius greater than the outer radius, an invalid object will result and be removed from the model.

5. Double click **CreateCylinder** under *ring_2* as shown in the following figure.

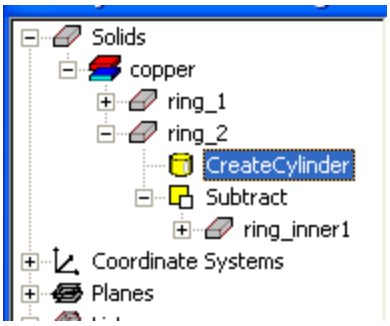


Figure 3-9 Command History Tree

The **Properties** dialog box appears.

6. Change **Radius** to: *0.5* in.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	0 ,0 ,0	in	0in , 0in , 0in
	Axis	Z		
	Radius	0.5	in	0.5in
	Height	5	in	5in
	Number of Seg...	0		0

Figure 3-10 Properties for ring_2

7. Click **OK**.

   In the structure, *ring_2* gets updated with the new value 0.5 inches of the outer radius.

8. Double click **CreateCylinder** under *ring_inner1.*

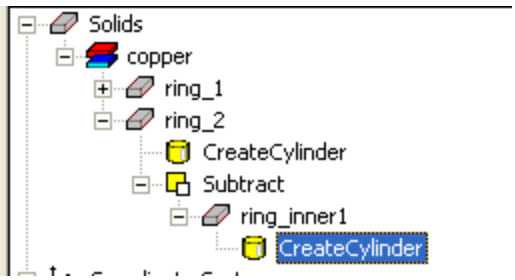


Figure 3-11 Command History Tree

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Properties** dialog box for *ring_inner1* appears.

9. Change the radius to: 0.435 in.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	0 ,0 ,0	in	0in , 0in , 0in
	Axis	Z		
	Radius	0.435	in	0.435in
	Height	5	in	5in
	Number of Seg...	0		0

Figure 3-12 Properties for ring_inner

10. Click **OK**.

The Modeler window shows two concentric rings.

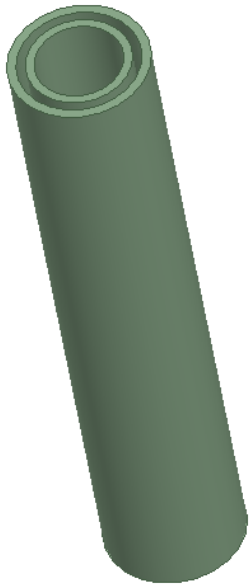


Figure 3-13 The conductor with the concentric rings

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Arm _1

The structure you are drawing in this project is a dipole antenna. Draw the conducting two arms. Draw the first arm 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 axis and drag the mouse to draw the rectangle.

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 to complete the box.

   The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-0.1 ,-0.31 ,5	in	-0.1in , -0.31in , 5in
	XSize	0.2	in	0.2in
	YSize	-4.69	in	-4.69in
	ZSize	-0.065	in	-0.065in

Figure 3-14 Properties dialog box for Arm_1

5. Edit the fields as shown in "Properties dialog box for Arm_1" above. .
6. Click the **Attribute** tab.
7. Type *Arm_1* in the **Name** field.

   Note: The conducting arm has copper as its material.

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

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

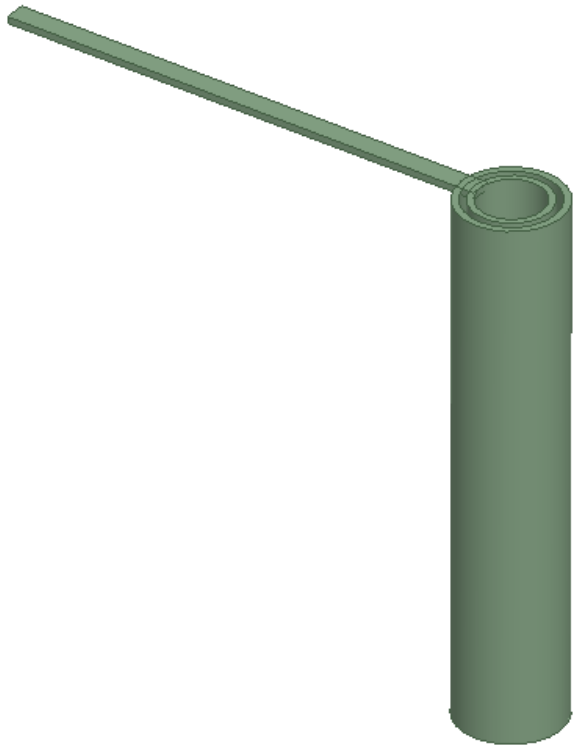


Figure 3-15 The Arm_1

Arm_1 is created.

Group the Conductors

You can now select all of the conductors drawn thus far to connect them together.

1. Click **Edit>Select All Visible** or press **CTRL+A**.
2. Click **Modeler>Boolean>Unite.**
   This unites the conductor and the Arm_1. See the following figure.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

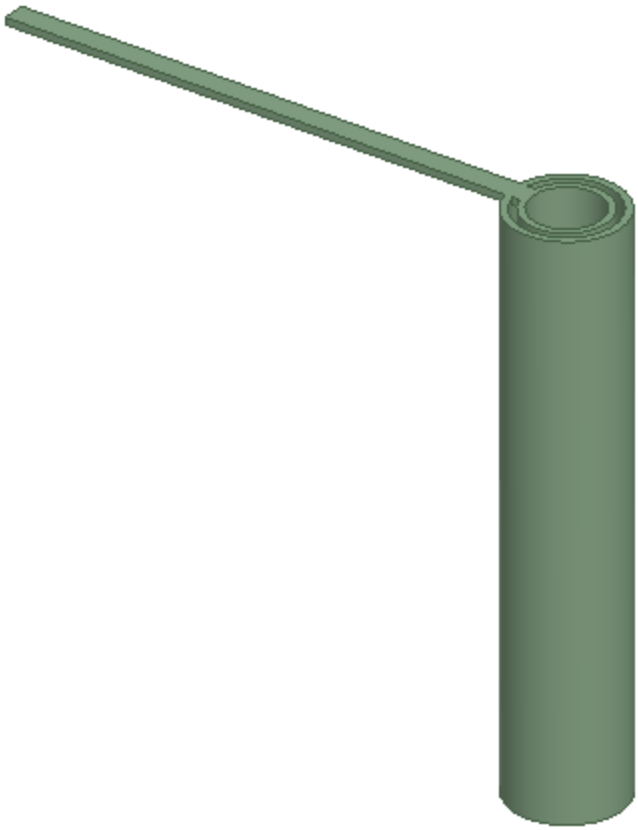


Figure 3-16 The conductor united with arm_1

Create the Center Pin

1. Draw a **Cylinder at the origin, with radius 0.1 inch and height = 5.1 inches.** The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate System	Global		
	Center Position	0 ,0 ,0	in	0in , 0in , 0in
	Axis	Z		
	Radius	0.1	in	0.1in
	Height	5.1	in	5.1in

Figure 3-17 Center Pin Properties

2. Edit the fields in the Command dialog box as in "Center Pin Properties" above. .
3. Click the **Attribute** tab.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Type *Center_pin* in the **Name** field.
5. Select Copper from the **Material** drop-down menu.
6. Click **OK**.

   The **Properties** dialog box closes and the box is updated with the new dimensions and properties.

   Note In the **Message Manager** window the same message related to Solve inside appears for *Center_pin.*

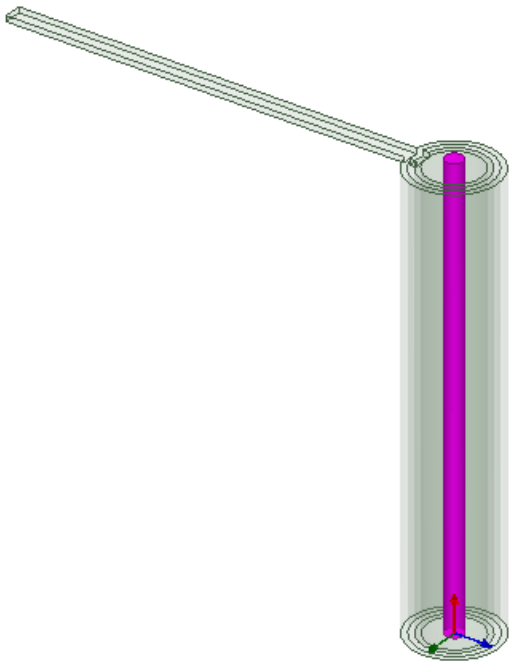


Figure 3-18 Center Pin placed inside the structure

Create Arm_2

1. Draw a Box freehand.

   The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate System	Global		
	Position	-0.1 ,0 ,5.1	in	-0.1in , 0in , 5.1in
	XSize	0.2	in	0.2in
	YSize	5	in	5in
	ZSize	-0.065	in	-0.065in

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3-19 Arm_2 Properties

2. Edit the fields in the **Command** dialog box as in "Arm_2 Properties" above. .
3. Click the **Attributes** tab.
4. Type: *Arm_2* in the **Name** field.
5. Click **OK**.

   The Properties dialog box closes and the box is updated with the new dimensions and properties.

   Note In the **Message Manager** window the same message related to SolveInside appears for *Arm_2.*
6. Click **View>Fit All>Active View**.

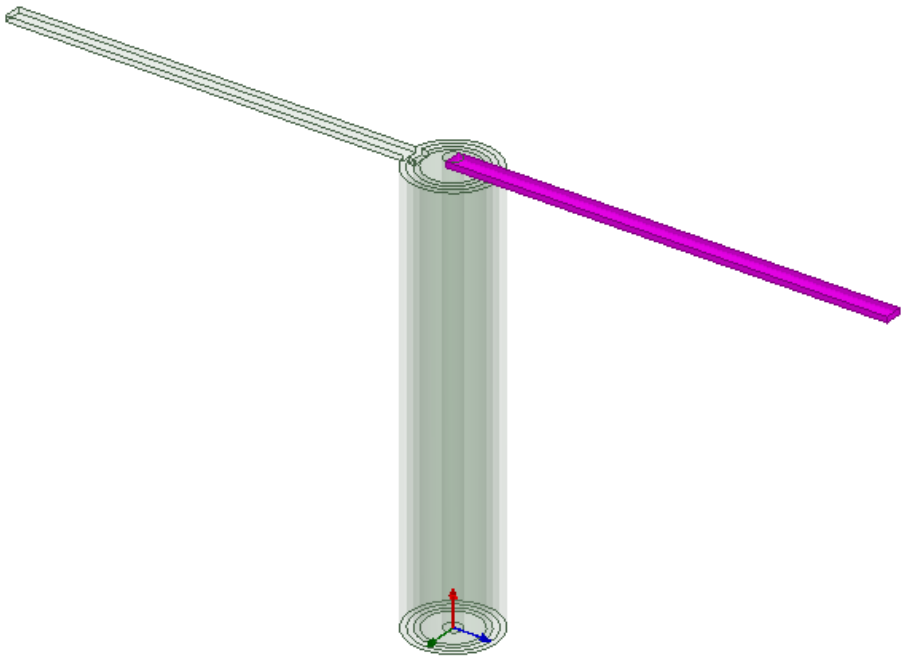


Figure 3-20 Dipole antenna with two conducting arms

Create the Grounding Pin

1. Draw a Cylinder freehand.

   The **Properties** dialog box appears.
2. Edit the fields of the **Command** dialog box as in the following figure.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate System	Global		
	Center Position	0 ,1 ,0	in	0in , 1in , 0in
	Axis	Z		
	Radius	0.0625	in	0.0625in
	Height	5.1	in	5.1in

Figure 3-21 Grounding pin properties

3. Click **Attribute** and then, type *pin* in the **Name** field.
4. Select *Copper* from the **Materials** drop-down menu.
5. Click **OK**.

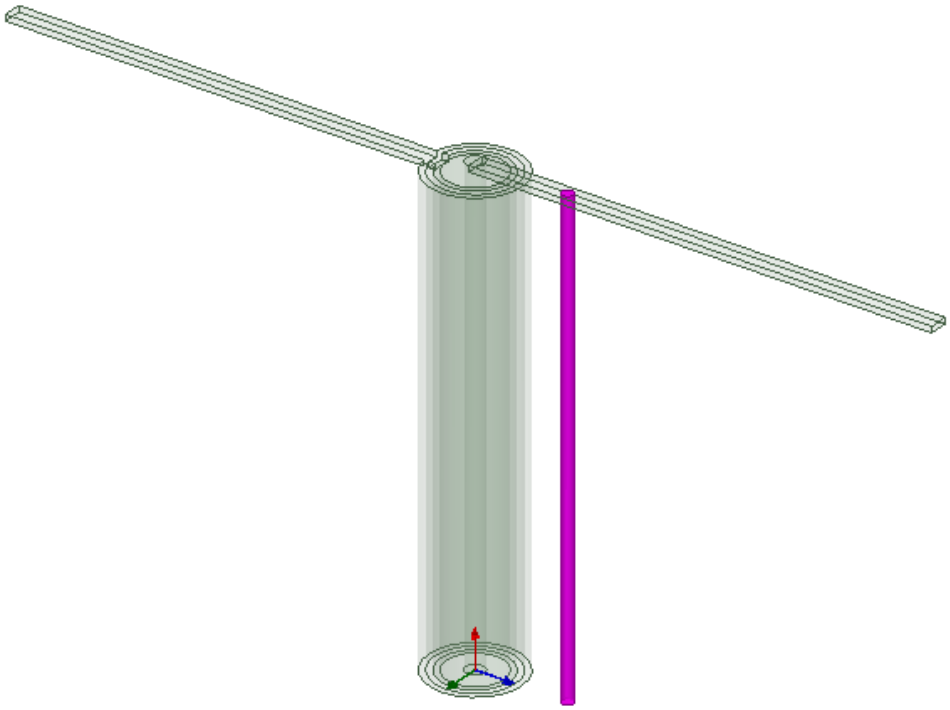


Figure 3-22 Grounding Pin Inserted

The Properties dialog box closes and the pin is updated with the new dimensions and properties.

Note In the **Message Manager** window the same message related to SolveInside appears for *pin.* Right-click in the Message Manager window and select “Clear all the messages. . .” from the short-cut menu.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Conductors

Unite the second arm, center pin, and the grounding pin.

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

   The **Select Object** dialog box appears.

2. Select the objects named: *Arm_2*, *center_pin*, and *pin*. See "Select Object dialog box" below. .

   Note Remember the order in which you select the objects. This is important because the order will determine which of the item(s) will appear in the subsequent dialog boxes. See **Note** below.

3. Click **OK**.

   The selected objects are highlighted in the modeler. See "Arm_2, Center Pin, and Grounding Pin selected" on the next page. .

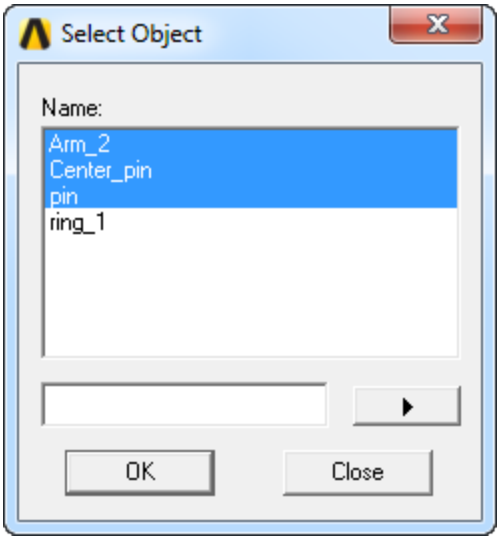


Figure 3-23 Select Object dialog box

4. Click **Modeler>Boolean>Unite**

   The 3 pieces are now united and the combined structure will appear as Arm_2 in the History Tree. See "Arm_2, Center Pin, and Grounding Pin selected" on the next page. and "The structure appears as Arm_2" on the next page. .

   Note You can also right click in the 3D modeler window and select **Edit**>**Boolean**>**Unite** from the short-cut menu.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

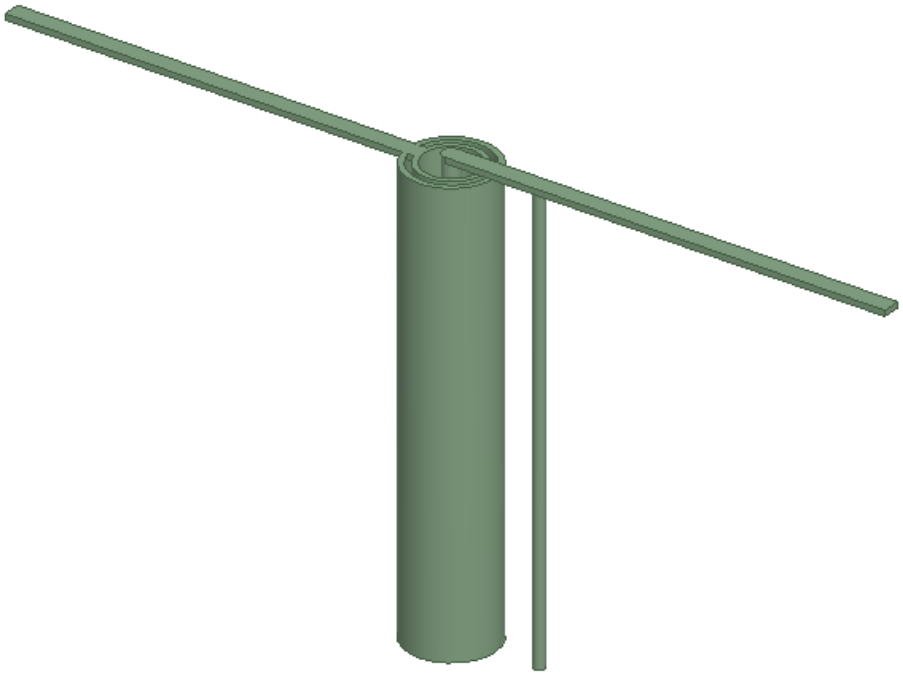


Figure 3-24 Arm_2, Center Pin, and Grounding Pin selected

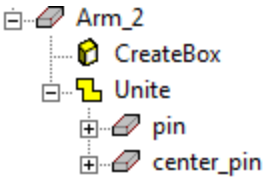


Figure 3-25 The structure appears as Arm_2

Note If you clicked *center_pin* first in the **Select Object** dialog box (Figure 23) and then united it with *pin* and *Arm_2*, then the combined structure's name will appear as *center_pin* on the **History Tree**.

Create a Circle

You will draw a circle and later assign a port to it. This circle is drawn such that it is the bottom face of the structure.

1. Click **Draw>Circle.**
2. Draw a circle and edit the fields in the Status Bars that appear at the bottom of the HFSS application.
   - Enter the center position: **X:***0.0*, **Y:***0.0*, **Z:***0.0*.
   - Press **Enter.**

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter the radius: **dX:***0.31*, **dY:***0.0*, **dZ:***0.0.*
- Press **Enter**.

The Properties dialog box appears.

3. Type *p1* in the **Name** field.
4. Click **OK**.

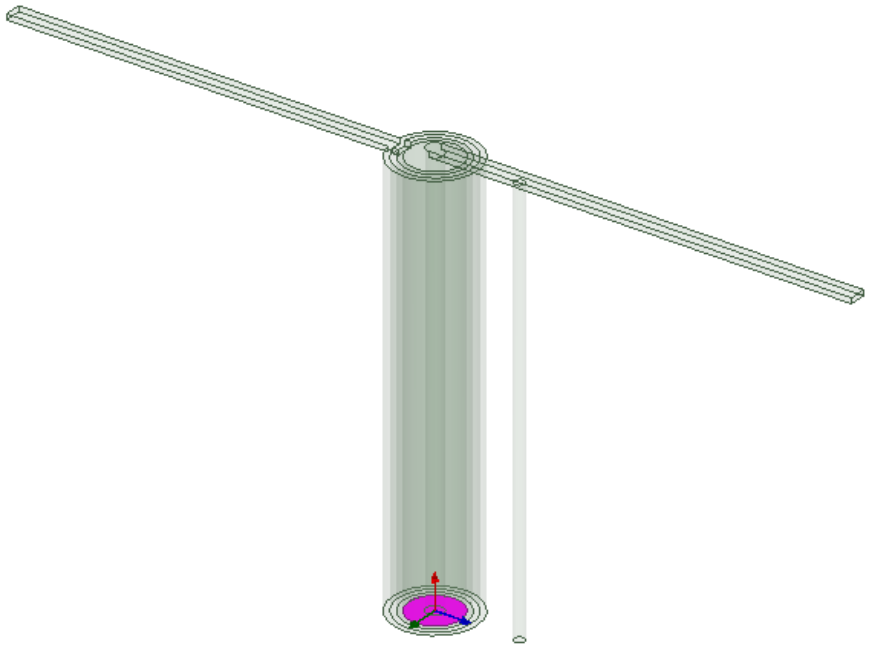


Figure 3-26 The Circle to Represent the port

Draw the Enclosure for Air

Before drawing the box from the 3-D Modeler toolbar, select vacuum.

1. Draw a box and enter the dimensions as in "Air Box properties" on the next page. .
2. Set the **Name** field to *Air*.
3. Check the **Display Wireframe** option. See "Attributes tab" on the next page. .
4. Click **OK**.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate System	Global		
	Position	-5 ,-10 ,0	in	-5in , -10in , 0in
	XSize	10	in	10in
	YSize	20	in	20in
	ZSize	12	in	12in

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

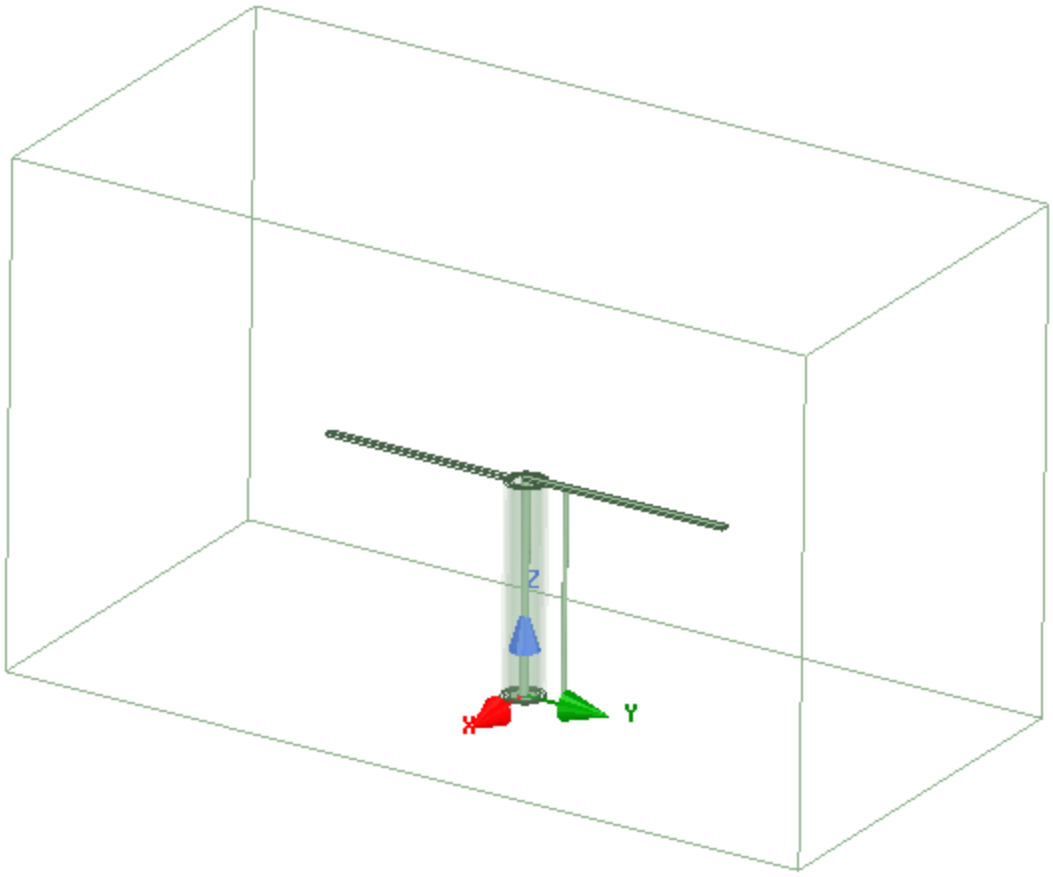
Figure 3-27 Air Box properties

	Name	Value	Unit	Evaluated Value
	Name	Box1		
	Material	"vacuum"		"vacuum"
	Solve Inside	☐		
	Orientation	Global		
	Model	☑		
	Display Wireframe	☑		

Figure 3-28 Attributes tab

The box with its wireframe now encloses the UHF-probe.

5. Do **Ctrl+D** to fit the view.
6. Click outside the box to view the wireframe and the structure it encloses.



ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3-29 Wireframe that encloses the probe

Create Radiation Boundary

1. Click **Edit>Select>Faces**
2. Click **Edit**>**Select By Name**.
3. Select *Air* from the **Select Face** dialog box.

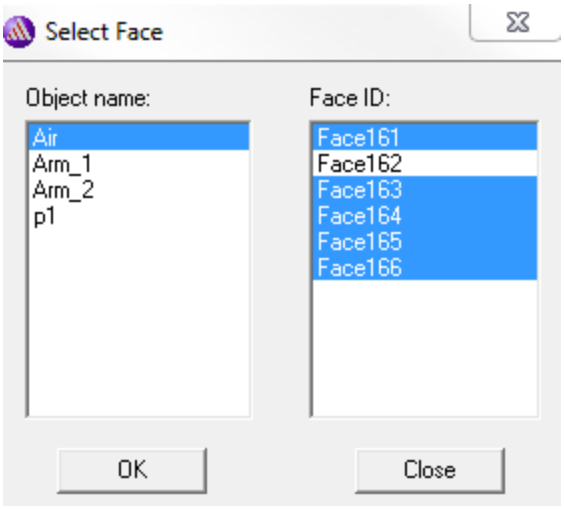


Figure 3-30 Select Face dialog box

4. Hold the **CTRL** key and select all of the 5 faces of the Air vacuum object except the face on the XY plane, i.e., the base (z=0).

   Note Each air face has a specific name and Face ID. This will help you differentiate between the faces as you select them. Especially, the bottom and the top surfaces.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

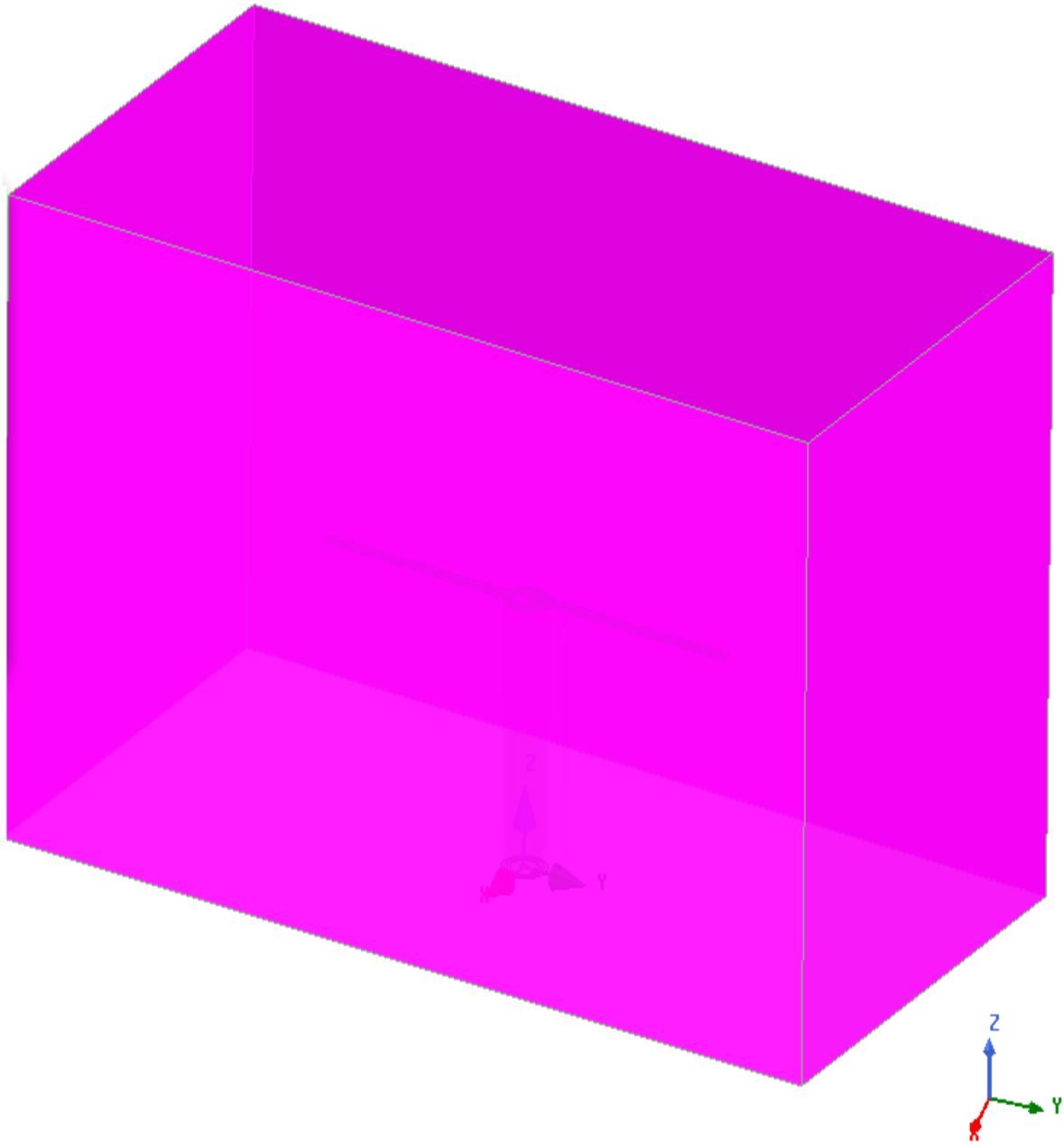


Figure 3-31 5 Faces selected except the bottom face

5. Right click inside the modeler and select **Assign Bounday>Radiation** from the short-cut menu.

   The **Radiation Boundary** dialog box appears.

6. Edit the fields as shown in the following figure.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

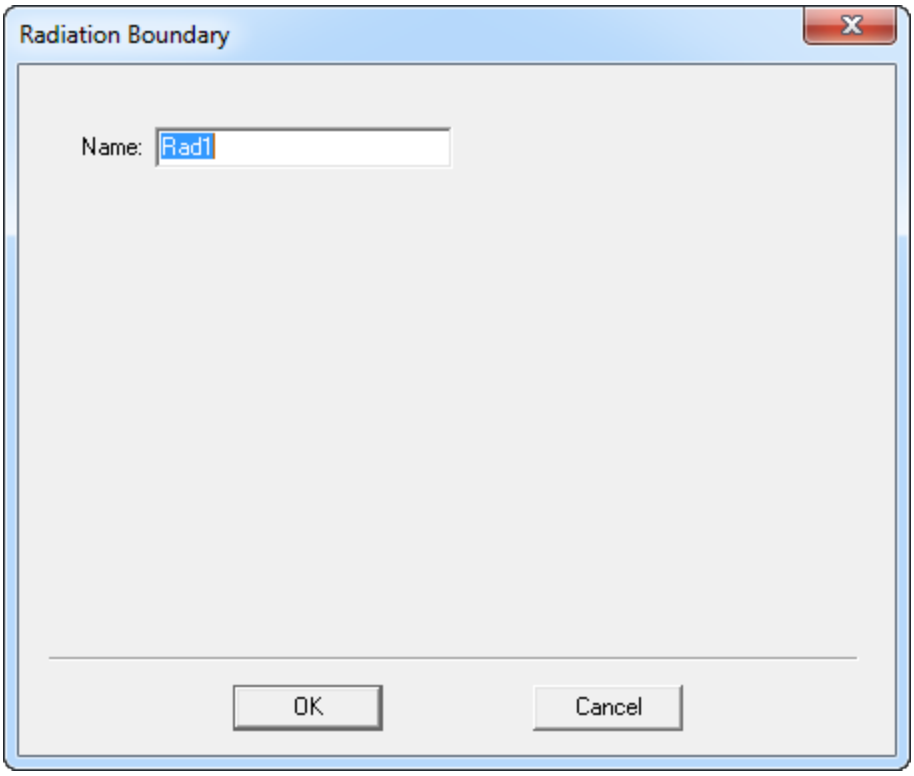


Figure 3-32 Radiation Boundary dialog box

7. Click **OK**.
   The radiation boundary is applied.

Assign Excitation

You can now assign wave ports after you select the bottom surface of the pin p1.

1. Click outside the box to view the structure in the wireframe.
2. Rotate the scroll wheel of the mouse in the counterclockwise direction to enlarge the view.
3. Click **Edit>Select>By Name**
   The **Select Face** dialog box appears.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

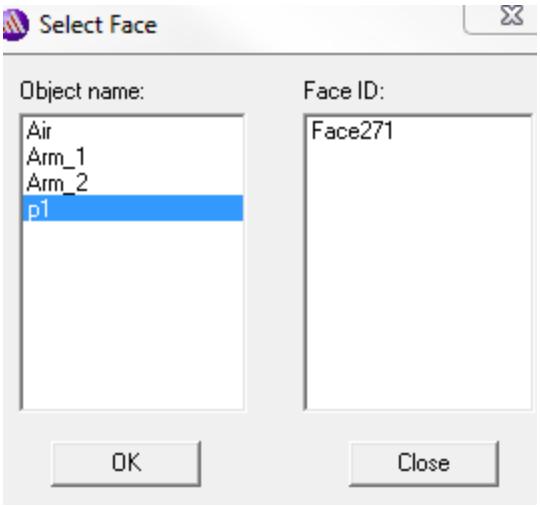


Figure 3-33 Select Face dialog box

4. Click *p1* and select the Face ID.
5. Click **OK**.
6. Right click inside the modeler window and select **Assign Excitation**>**Wave Port from the short-cut menu.**

   The **Reference Conductors for Terminals** dialog box appears.
7. Check *Arm_1* or *ring_1*

   Note It really depends upon the order you selected the solids before uniting them that determines what will appear on the **Reference Conductors for Terminals** dialog box. For instance, Figure 34 illustrates that *Arm_1* was selected first before uniting it with *ring_1* and hence, *Arm_1* appears on the dialog box of **Reference Conductors for Terminals**. Figure 35 illustrates an identical model but when drawn *ring_1* was selected first and then, united with *Arm_1* so *ring_1* appears on the dialog box. Therefore, the order in which you select the items before uniting determines which item holds precedence over the other in its appearance on the dialog box.
8. Click **OK**.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

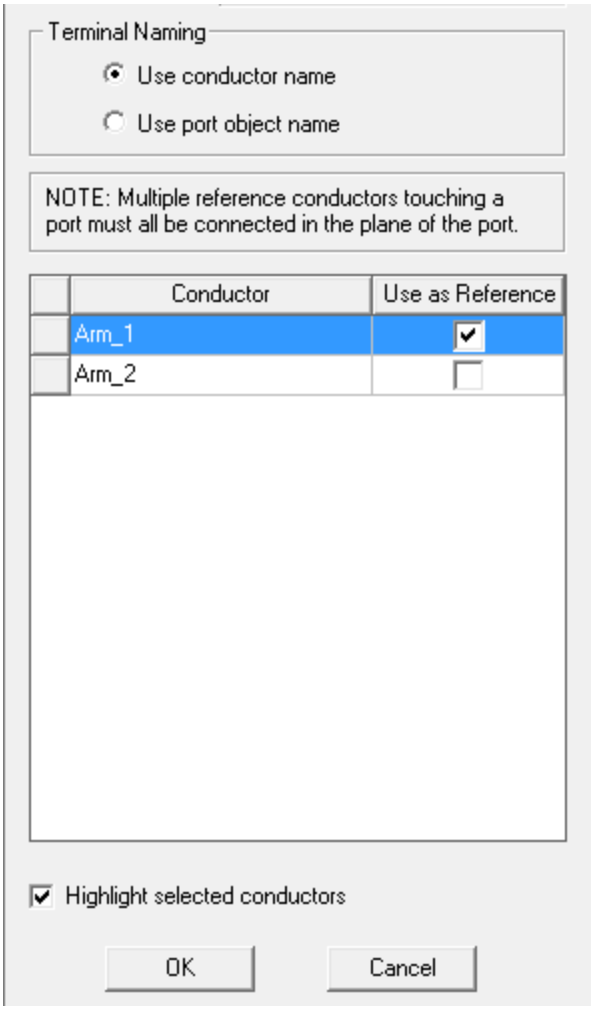


Figure 3-34 Reference Conductors for Terminals dialog box

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

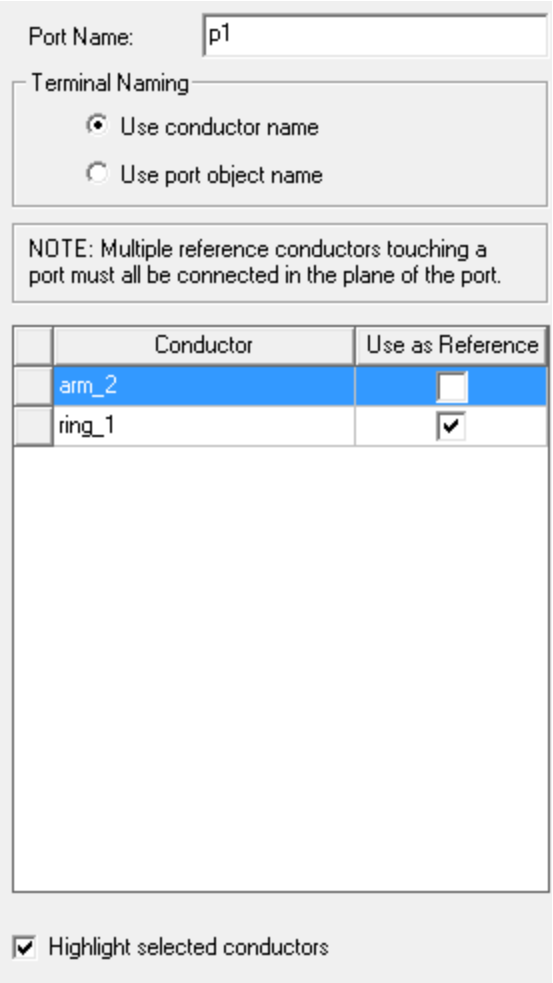


Figure 3-35 Reference Conductors for Terminals

Rename the Wave Port and Terminal

We will consider the case for which the dialog box shown in "Terminal dialog box" on the facing page. appears before we proceed to edit the waveport and terminals.

1. Double-click *1* under **Excitations**.
   The **Wave Port** dialog box appears.
2. Type *p1* in the **Name** field and click **OK**.
3. Double-click the terminal named *Arm_2_T1.*
   The **Terminal** dialog box appears.
4. Enter *T1* in the **Name** field.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

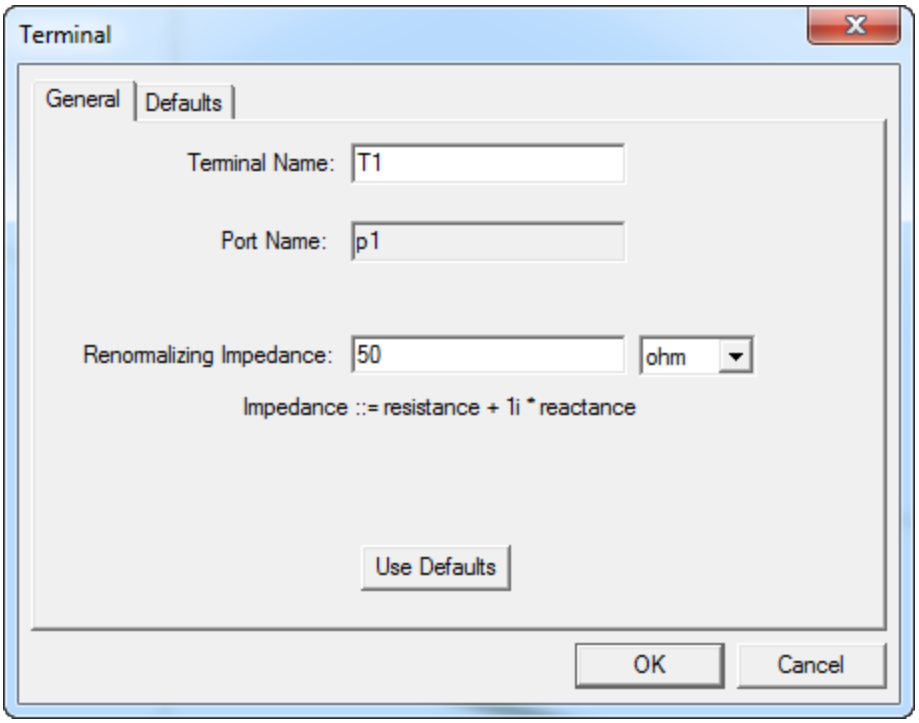


Figure 3-36 Terminal dialog box

This renames the terminal to T1.

5. Click **OK**.

Create Infinite Ground Plane

1. Click **Edit**>**Select**>**Faces.**
2. Select the bottom face of the Air object at Z=0.

   Note Recall this was the only face on which we did not apply the radiation boundary.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

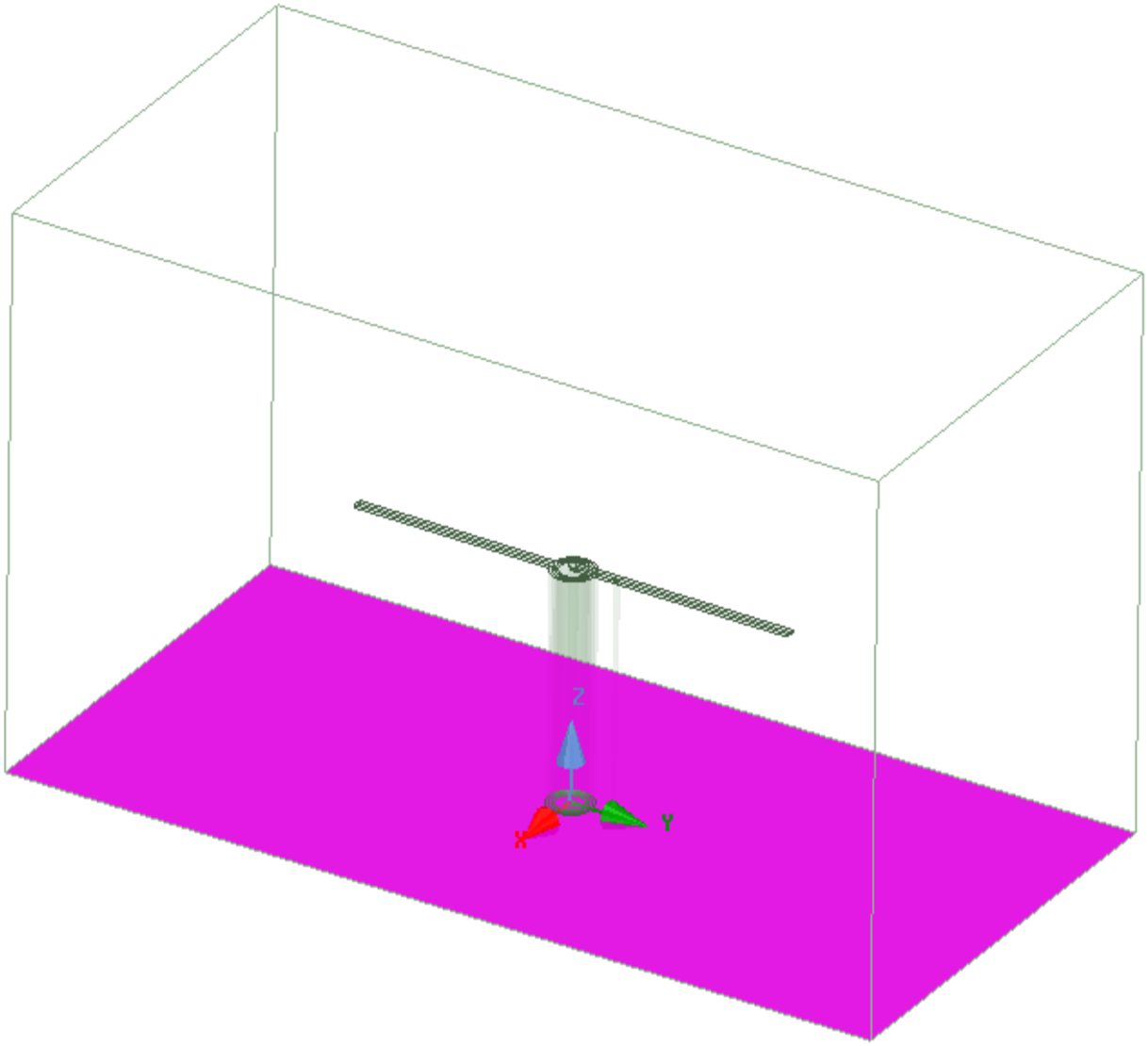


Figure 3-37 The bottom face selected

3. Click **HFSS>Boundaries>Assign> Finite Conductivity**
   The **Finite Conductivity Boundary** dialog box appears.
4. Set the fields as shown in "Finite Conductivity Boundary" on the facing page. .
5. Click the **vacuum** button.
   The **Select Definition** dialog box appears**.**
6. Type *copper* in the **Search by Name** field.
7. Click **OK**.
8. Click **OK** to close the **Finite Conductivity Boundary** dialog box.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The boundary gets assigned on the bottom face. See "Finite Conductivity Boundary applied" on the next page. .

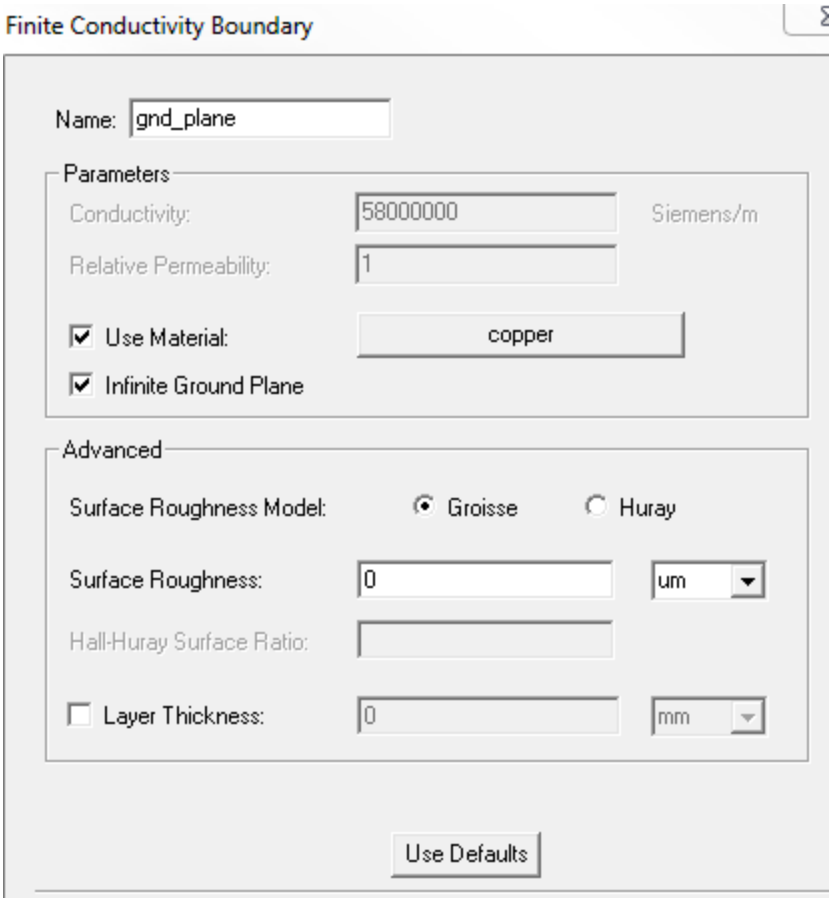


Figure 3-38 Finite Conductivity Boundary

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

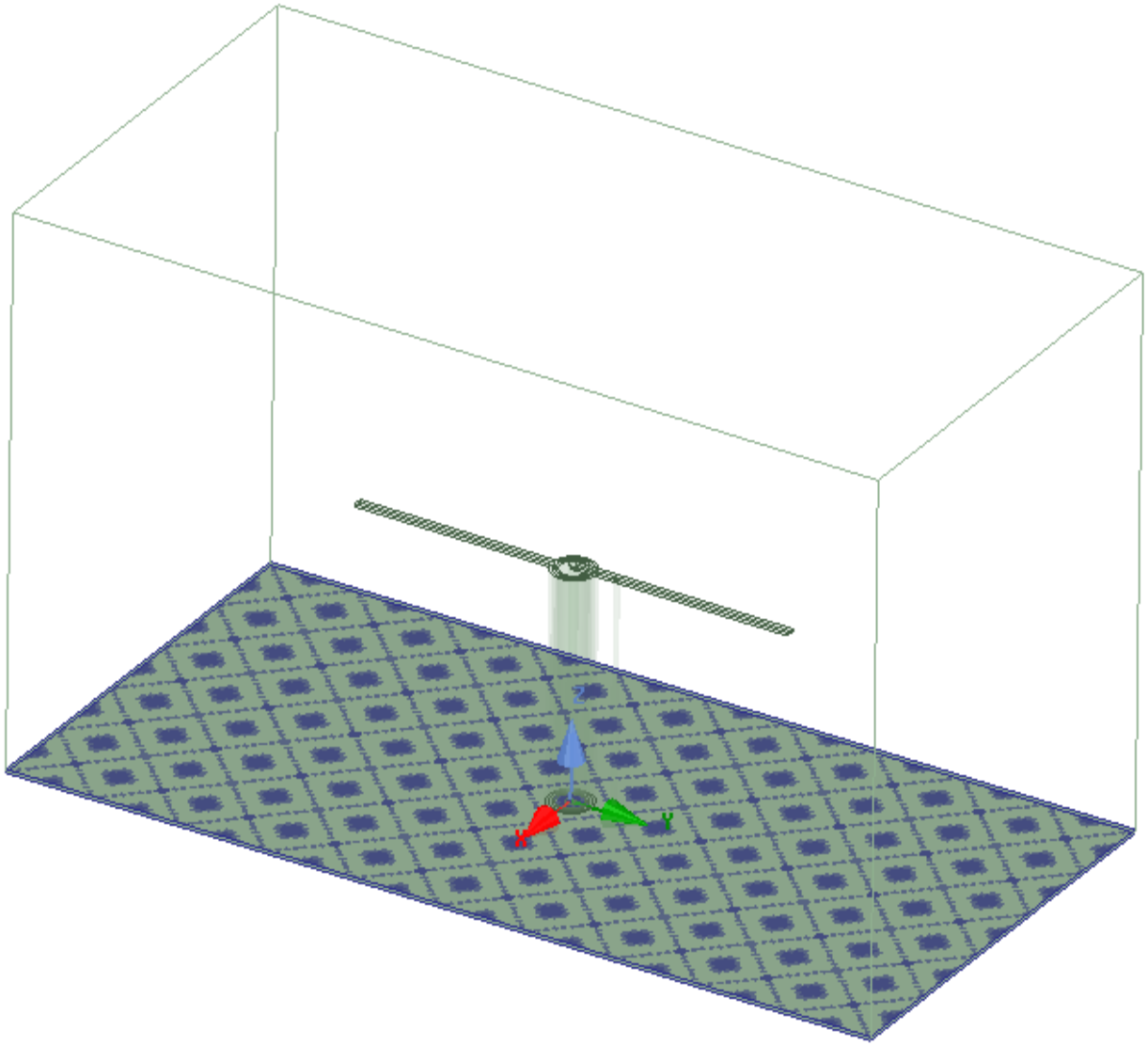


Figure 3-39 Finite Conductivity Boundary applied

Create a Radiation Setup

1. Click **HFSS>Radiation>Insert Far Field Setup>Infinite Sphere**
   The **Far Field Radiation Sphere Setup** dialog box appears.
2. Edit the fields as shown in "Far Field Radiation Sphere Setup dialog box" on the facing page.
   .
3. Click **OK**.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note The **Far Field Radiation Sphere Setup** dialog box closes and the **ff_2d** option appears under **Radiation** in the **Project Manager** window.

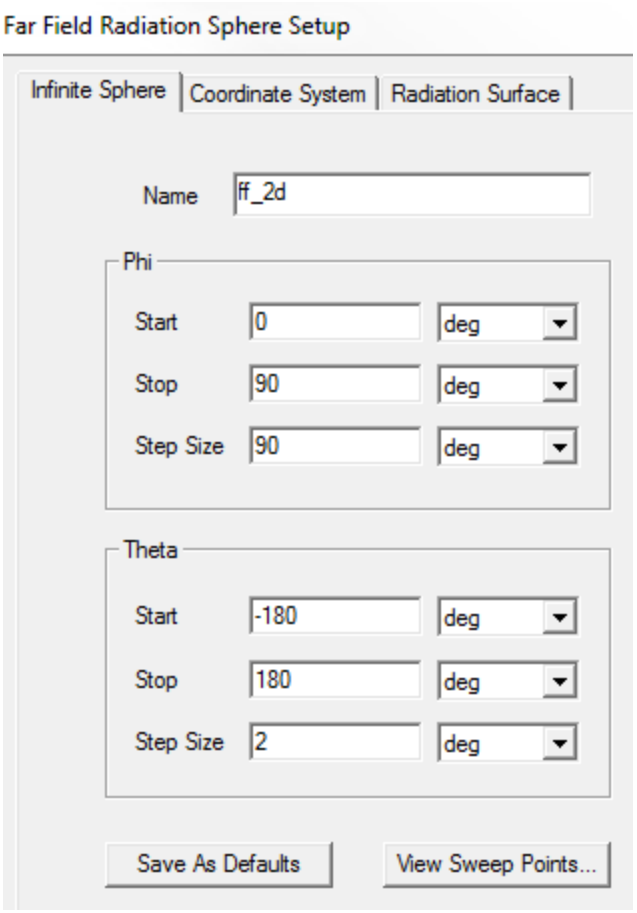


Figure 3-40 Far Field Radiation Sphere Setup dialog box

Add Length Based Mesh Operation to the Radiation Boundary

Far fields are calculated by integrating the fields on the radiation surface. To obtain accurate far fields for antenna problems, the mesh size on the integration surface should be on the order of λ/6 to λ/8 maximum /tetrahedra length. Mesh size refers to the lengths of the edges of the surface mesh.

To create a length based seed on the radiation boundary:

1. Press the **O** key to enter Object Selection mode.
2. Select **Air** from the history tree to highlight the enclosure.
3. In the Project Tree right click **Mesh Operations>Assign>On Selection>Length Based**.

   The **Element Length Based Refinement** dialog box appears.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set the fields as in "Element Length Based Refinement" below. .
5. Enter *3.5 in* in the **Maximum Length of Elements:** field

   Note This is about λ/6 at 0.55GHz.

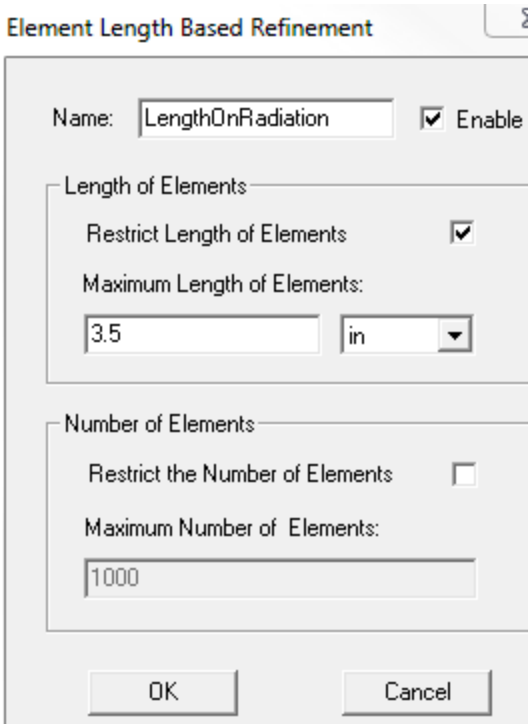


Figure 3-41 Element Length Based Refinement

6. Click **OK**.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Analyze the Model

This chapter contains the following topics:

- Launch HFSS and Set up the Project
- Launch HFSS
- Set Tool Options
- Open and Save a New Project
- Set the Model Units

Create an Analysis Setup

1. Right click **Analysis** from the Project Tree and select **Add Solution Setup** from the short-cut menu.

   The **Solution Setup** dialog box appears.

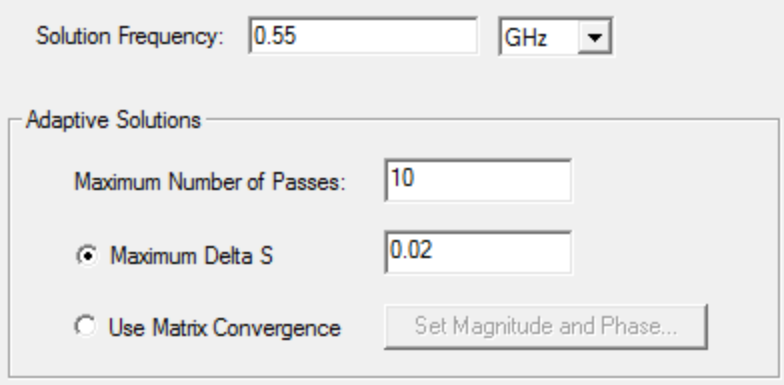


Figure 4-1 Solution Setup dialog box

2. Click **General**.
3. Edit the fields as shown in "Solution Setup dialog box" above. .
4. Click the **Options** tab.
5. Check **Enable Iterative Solver**.
6. Click **OK**.

Add Frequency Sweep

1. Right click **Setup1** under **Analysis** and select **Add Frequency Sweep from the short-cut menu.**

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

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Edit the fields as shown in "Edit Frequency Sweep dialog box" below. .
3. Click **OK**.

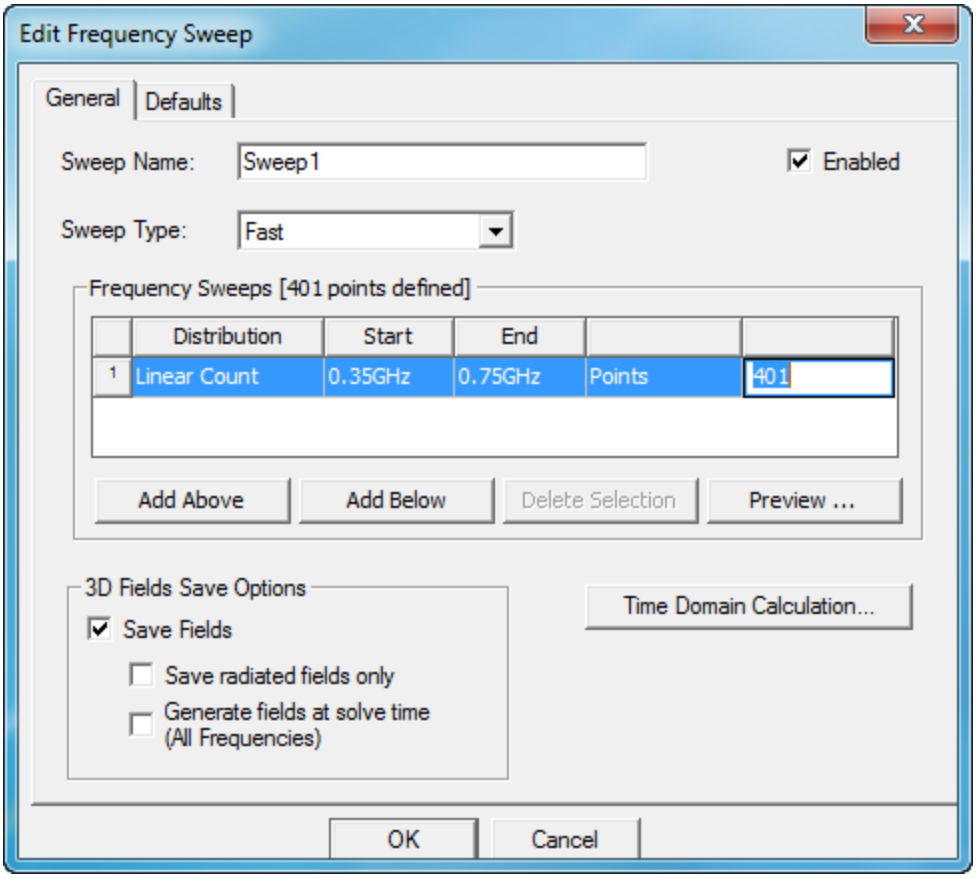


Figure 4-2 Edit Frequency Sweep dialog box

Validate Model

Before you run the simulations, your design must pass the validate check.

1. Click **HFSS>Validation Check**

   The **Validation Check** dialog box appears.



ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4-3 Validation Check dialog box

The **Validation Check** dialog box must show a tick near each of the listed options, in which case, you can analyze the design.

2. Click **Close**.

Note: For most HFSS projects, do not be alarmed if you see any warnings in the **Message Manager** window. Some of these messages warn you of potential problems. It may not always require you to perform any action to deal with the messages. In this project you noticed a few messages related to SolveInside earlier in the design stage of the project and you cleared those messages.

Analyze the UHF Probe

1. Right click **Analysis and select Analyze All from the short-cut menu.**

Note If you haven't saved the project, HFSS will prompt you to do so. After you save the project, HFSS will run the simulation and solve the design of the UHF Probe.

Normal completion of simulation occurs on the server.

View Solution Data

1. Click **HFSS>Results>Solution Data**

The **Solution Data** dialog box opens.

2. Click the **Profile** tab to view the solution profile.

Note You can view the synopsis of the simulation results ranging from information about the different Adaptive Passes, the matrix assembly for the S-parameters etc to various tasks that HFSS ran during the simulation.The number of tetrahedra increases with each adaptive pass tending to a near-accurate solution. You can alter the values of Max Delta S and Maximum Number of Passes and run HFSS to attain more accurate solutions but be careful that you spend computational resources judiciously. You do not want too dense a mesh neither do you want the mesh to be too coarse. Therefore, for a good quality solution use the values that we recommend as a guideline and modify them as needed.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Pre	00:00:06	00:00:06	71.4 M	1282 triangles
Mesh TAU (Strict)	00:00:03	00:00:03	88.9 M	30697 tetrahedra
Mesh TAU (Coarseni...	00:00:02	00:00:02	88.9 M	11572 tetrahedra
Mesh Post	00:00:00	00:00:00	90.5 M	11572 tetrahedra
Mesh Refinement				Lambda Based
Mesh (lambda based)	00:00:01	00:00:01	70.3 M	11080 tetrahedra
Mesh Refinement				Manual Seed Based
Mesh (volume, seed)	00:00:01	00:00:01	71.4 M	12385 tetrahedra
				LengthOnRadiation
Mesh Refinement				Port Adapt
Simulation Setup	00:00:00	00:00:00	46.1 M	Disk = 0 KBytes
Port Adaptation	00:00:00	00:00:00	55.6 M	Disk = 5 KBytes, 7546 tetrahedra
Mesh (port based)	00:00:02	00:00:02	71.2 M	12495 tetrahedra
Adaptive Pass 1				Frequency: 0.55 GHz
Simulation Setup	00:00:00	00:00:00	46.4 M	Disk = 0 KBytes
Matrix Assembly	00:00:01	00:00:00	75.4 M	Disk = 37 KBytes, 7654 tetrahedra , p1: 113 triangles
Solver DCS1-L2	00:00:03	00:00:03	111 M	Disk = 0 KBytes, matrix size 57841 , matrix bandwidth 19
Field Recovery	00:00:00	00:00:00	111 M	Disk = 2977 KBytes, 1 excitations
Adaptive Pass 2				Frequency: 0.55 GHz
Mesh (volume, adapti...	00:00:01	00:00:01	72.6 M	14798 tetrahedra
Simulation Setup	00:00:00	00:00:00	47.6 M	Disk = 0 KBytes
Matrix Assembly	00:00:01	00:00:01	83.3 M	Disk = 0 KBytes, 9649 tetrahedra , p1: 113 triangles
Solver DCS1-L2	00:00:02	00:00:02	125 M	Disk = 0 KBytes, matrix size 71011 , matrix bandwidth 20
Field Recovery	00:00:00	00:00:00	125 M	Disk = 997 KBytes, 1 excitations

Figure 4-4 Solution Profile

3. Click **Convergence** to view the convergence data.

   Note The default view is for convergence is Table.

4. Select **Plot** to view a graphical representations of the convergence data.
5. Set the Convergence fields as in the following figure.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

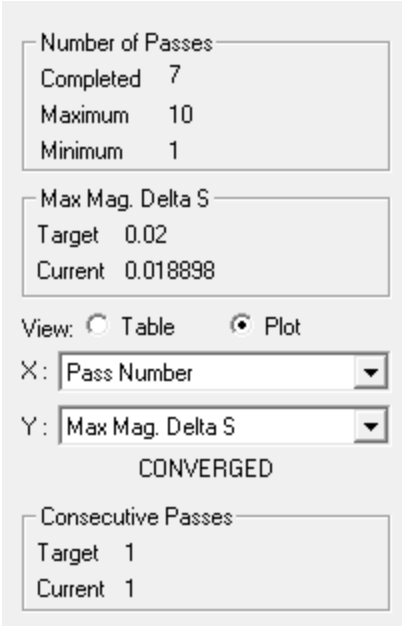


Figure 4-5 Convergence tab

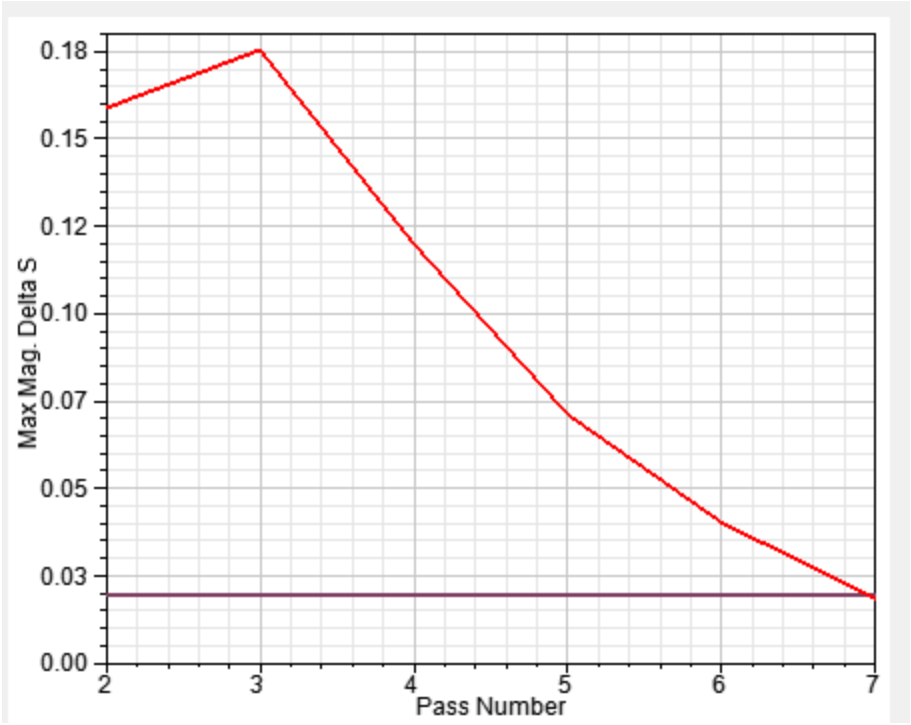


Figure 4-6 Convergence graph

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note At the 7th pass, the Maximum Delta S converges. It reaches a high at the 3rd pass and then, dip down to the value of 0.03 and then tends to 0.02 that you set at the 7th pass.

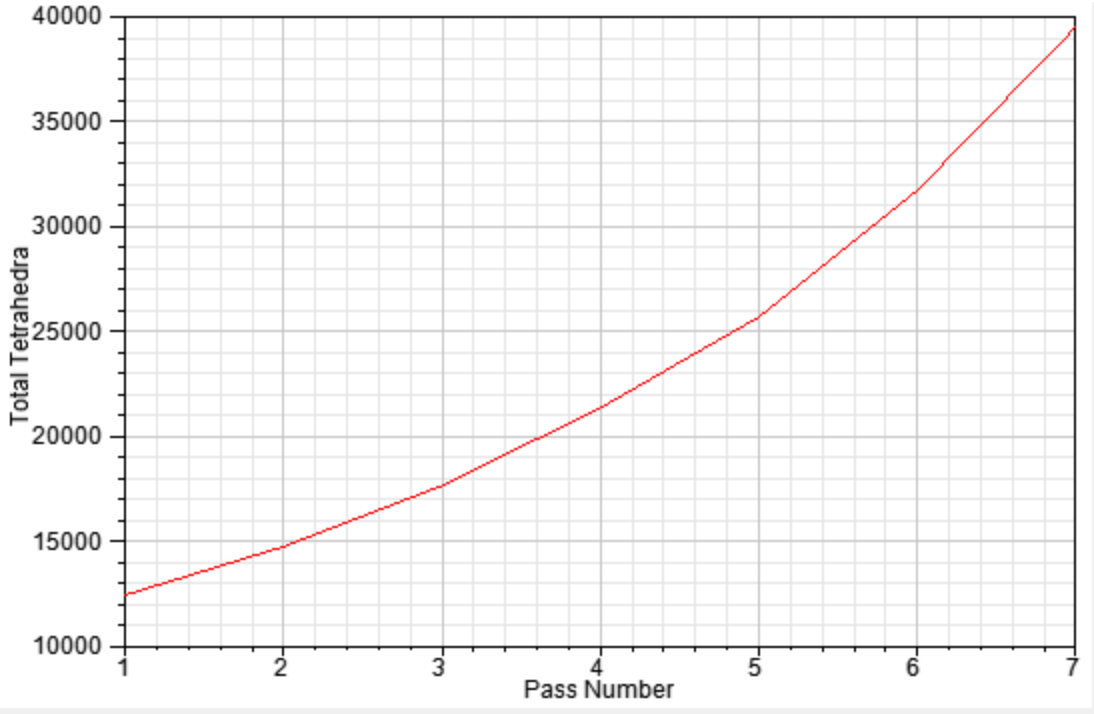


Figure 4-7 Total Tetrahedra with respect to Pass Number

Note Viewing this graph in "Total Tetrahedra with respect to Pass Number" above. clearly indicates and also confirmed from the Profile tab that as the pass number increases the number of tetrahedra solved for the mesh also increases.

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

6. Click **Mesh Statistics** to view the Mesh statics.
7. Click **Close**.

Create Reports

These sections will help you create reports and further analyze the design of the UHF probe.

Terminal S Parameter Vs Frequency

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

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

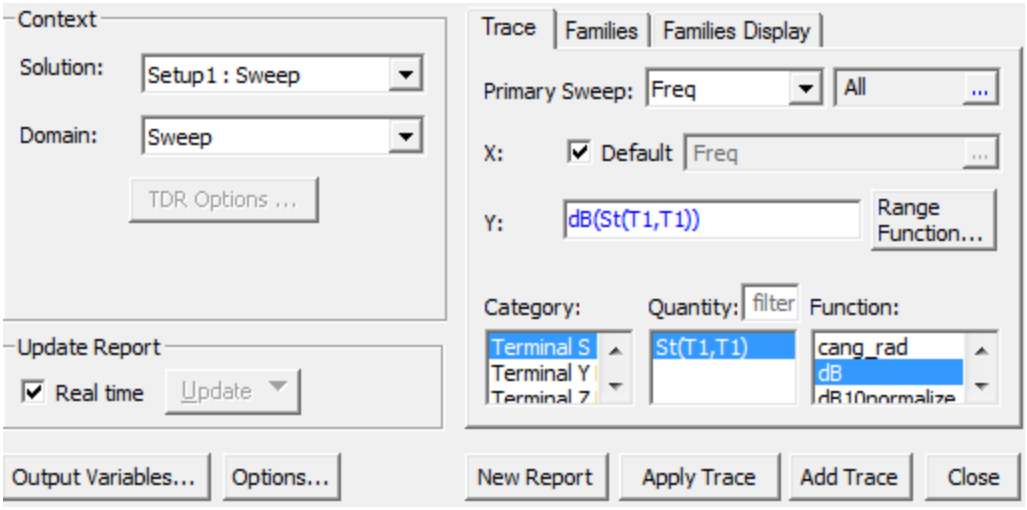


Figure 4-8 Report dialog box

2. Ensure the default settings are as in "Report dialog box" above. .
3. Click **New Report**.

   Note The plot is generated.

4. Click **Close**.

   Now you can see the plot clearly. See "S Parameter Vs Frequency" on page 4-9. .

Change the X-axis Label Settings

1. Double click the plot's x-axis.

   The **Properties** dialog box appears.

2. Change **Number Format** from *Auto* to *Decimal*.

   Note You may need to scroll down to see the properties to edit.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Change **Field Precision** from *2* to *0*

	Name	Value	Description
	Display Name	☑	
	Specify Name	☐	
	Name	Freq	
	Axis Color		
	Text Font	Font	
	Display Units	☑	
	--Window		
	Window Mode	Axis range	
	Window width ...	400	
	--Manual Format		
	Number Format	Decimal	
	Field Width	5	
	Field Precision	0	

Figure 4-9 Properties

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

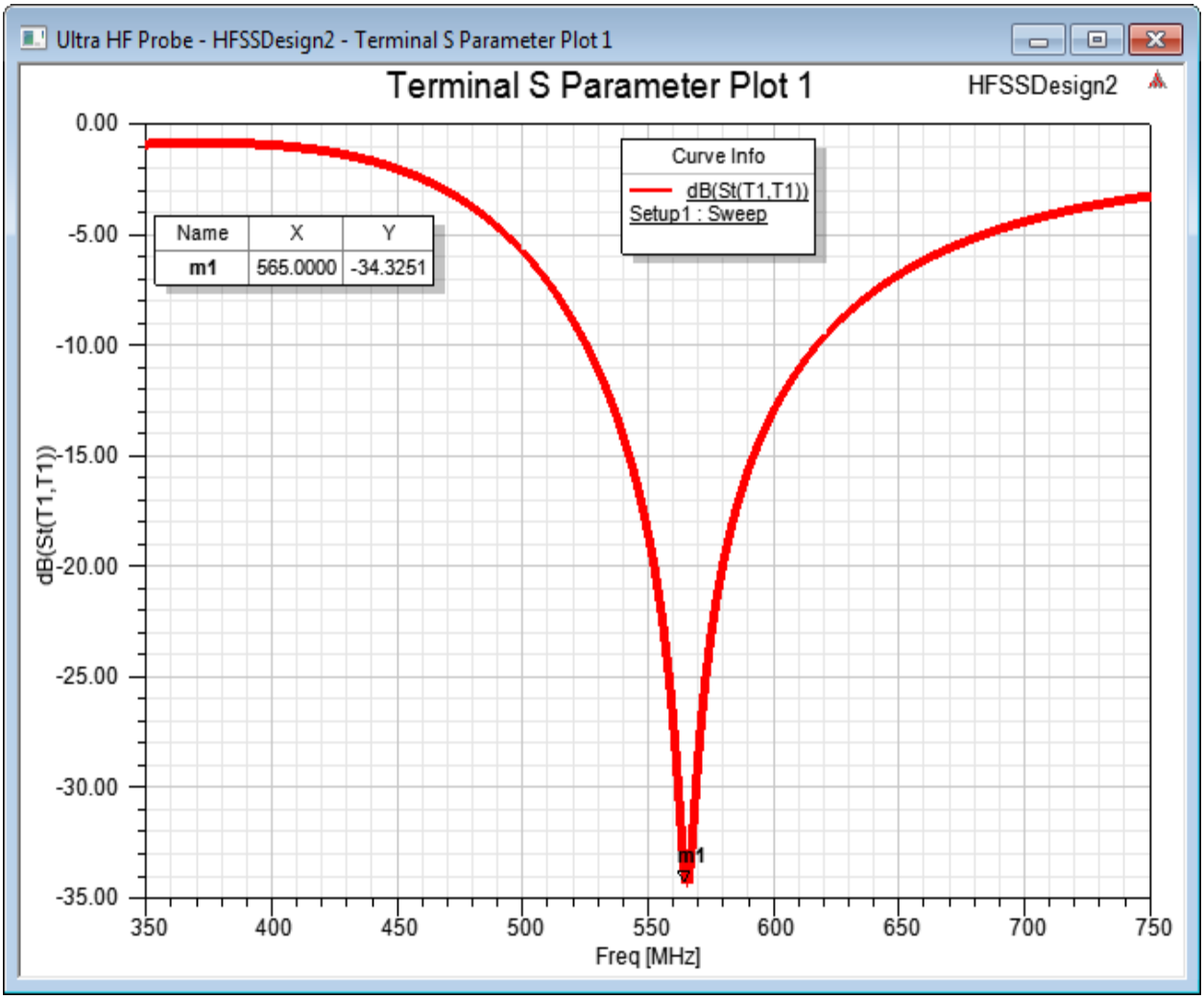


Figure 4-10 S Parameter Vs Frequency

5. Right-click the curve and go to **Marker > Add Minimum** from the short-cut menus to locate the resonant frequency of the ultra high frequency probe.

Create Far Field Overlay

To create a 2D polar far field plot:

1. Click **HFSS>Results>Create Far Fields Report>Radiation Pattern.**
   The **Report** dialog box appears.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

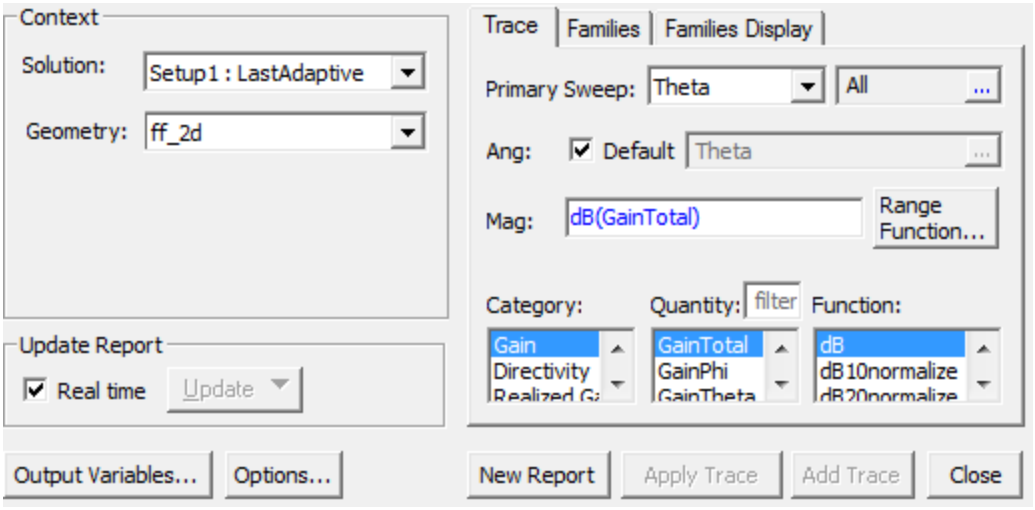


Figure 4-11 Report dialog box

2. Edit the fields as shown in "Report dialog box" above. .
3. Click **New Report**.
   The new report is displayed.
4. Click **Close**.

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

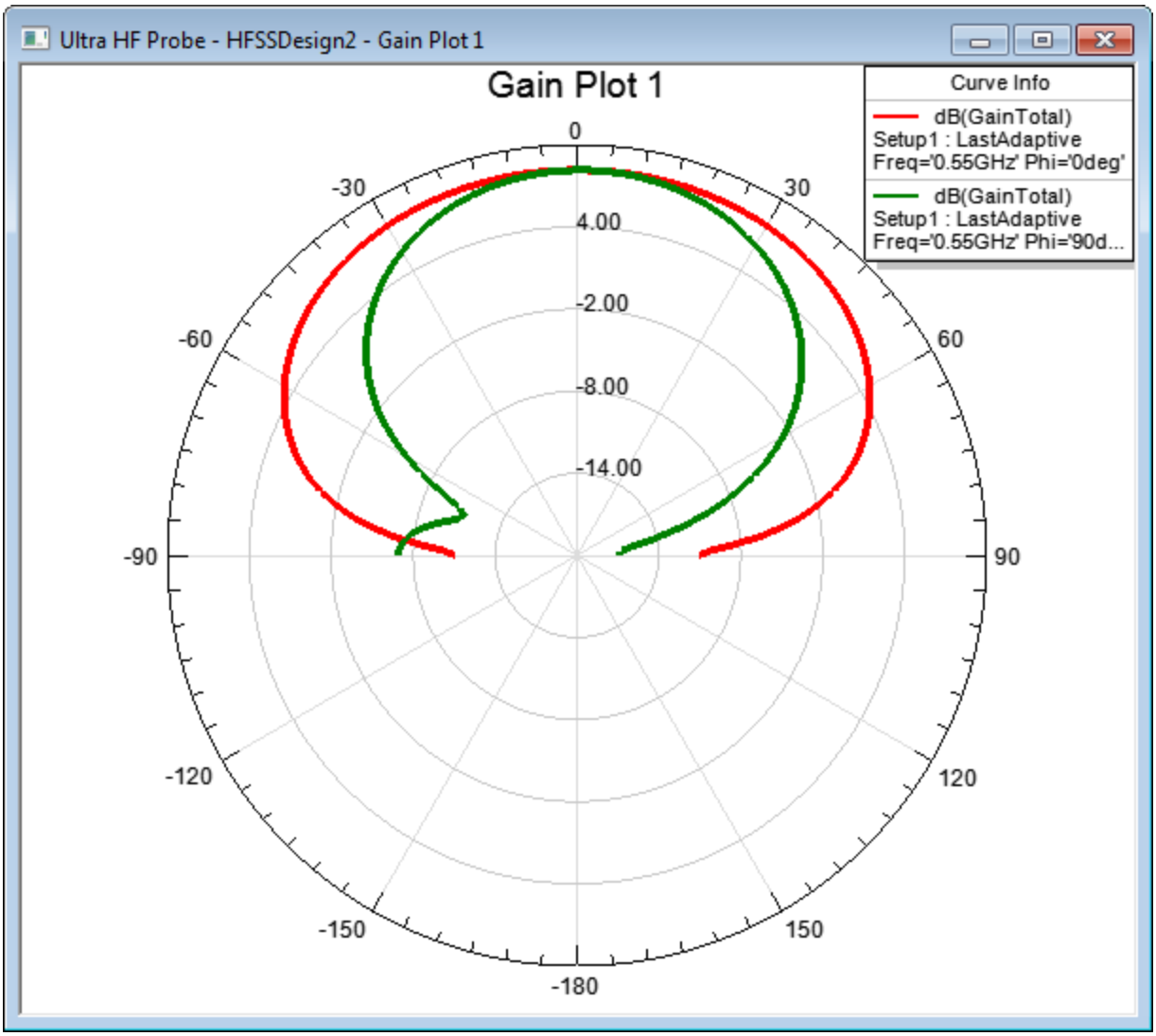


Figure 4-12 Radiation Pattern

ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.