

Getting Started with HFSS: Bandpass Filter

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**
HFSS Design Environment 1-2
2 - Set Up the Project **2-1**
Launch ANSYS Electronics Desktop 2-1
Set Tool Options 2-1
Insert HFSS design 2-3
Set Model Units (in) 2-4
Set Solution Type (Terminal) 2-5
3 - Create the 3D Model **3-1**
Create the 3-D Model 3-1
Create the Air Box 3-1
Create Feed1 (Coax Outer Diameter) 3-5
Create Feed Pin1 (Coax Inner Diameter) 3-6
Create Coax Feed Probe1 3-8
Create the Resonators 3-9
To create l1: 3-9
To create l2: 3-10
To create l3: 3-11
To create l4: 3-12
Assign Excitation 3-13
Create Rest of the Model by Duplication 3-16
Boundary Display (Optional) 3-19
4 - Analyze the Model **4-1**
Add HPC Analysis Setup 4-3
Analyze the Bandpass Filter 4-9
Review Solution Data 4-10
Review the Profile Panel 4-10

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

Review the Convergence Panel 4-12
Review Matrix Data Panel 4-13
Review Mesh Statistics Panel 4-14
Create Reports 4-14
Create S-Parameter vs. Frequency 4-14
Compare S12 with S21 4-16
Change Plot Scale 4-17
Create Field Overlays 4-19
Modify Plot Attributes 4-20

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

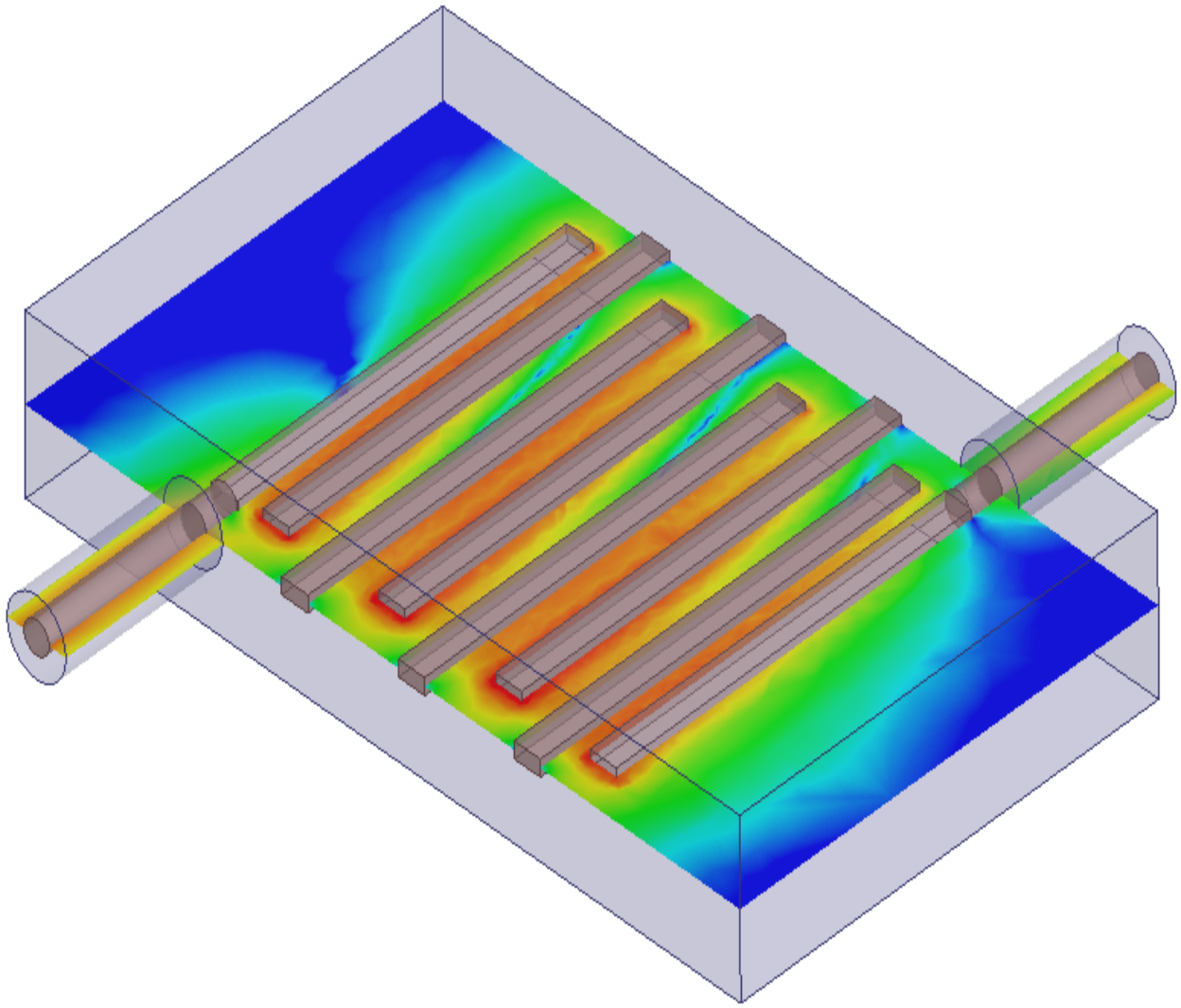
This document is intended as supplementary material to HFSS for beginners and advanced users. It includes instructions to create, solve, and analyze a bandpass filter.

This chapter contains the following topics:

- Sample Project - Bandpass Filter
- HFSS Design Environment

Sample Project - Bandpass Filter

A bandpass filter allows frequencies of a certain range to pass through and attenuate those outside it. "The Bandpass Filter Model" on the next page. is an illustration of a model representing bandpass filter. This tutorial describes how to draw this model.



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

Figure 1-1 The Bandpass Filter Model

HFSS Design Environment

There are many features in HFSS that enable you to create this model. Some of these features are listed below:

- Three-dimensional (3-D) solid modeling.
- Primitives such as cylinders and boxes
- Boolean operations such as duplicate around axis
- Boundaries and excitations
- wave ports and terminals
- design validation
- Solution setup and frequency sweep
- S-parameter plots.
- Field Overlays to plot electromagnetic fields
- Two-dimensional (2-D) and 3-D Field Plots

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 includes the following sections:

- Launch ANSYS Electronics Desktop
- Set Tool Options
- Insert HFSS design
- Set the Model Units
- Set the Solution Type

Launch ANSYS 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.

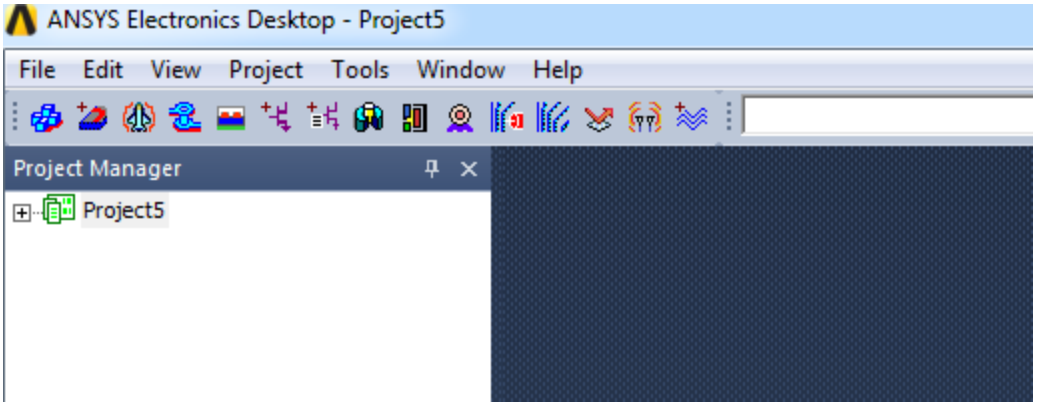


Figure 2-1 ANSYS Electronics Desktop launched

Note If the project folder is missing, go to **File**and click **New** to start a new project. If the **Project Manager** window does not appear, go to **View** and enable it.

Set Tool Options

Before you begin creating a design, configure some of the general options of HFSS:

1. Click **Tools>Options>General Options.**

   The **Options** dialog box appears.

2. Expand **HFSS** and select **Boundary Assignment**.

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

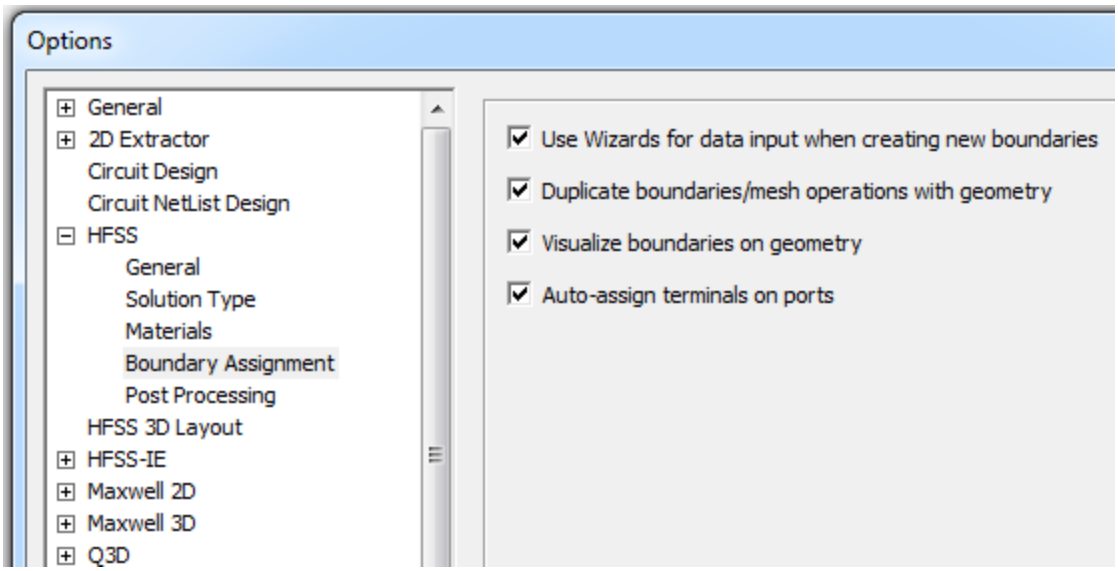
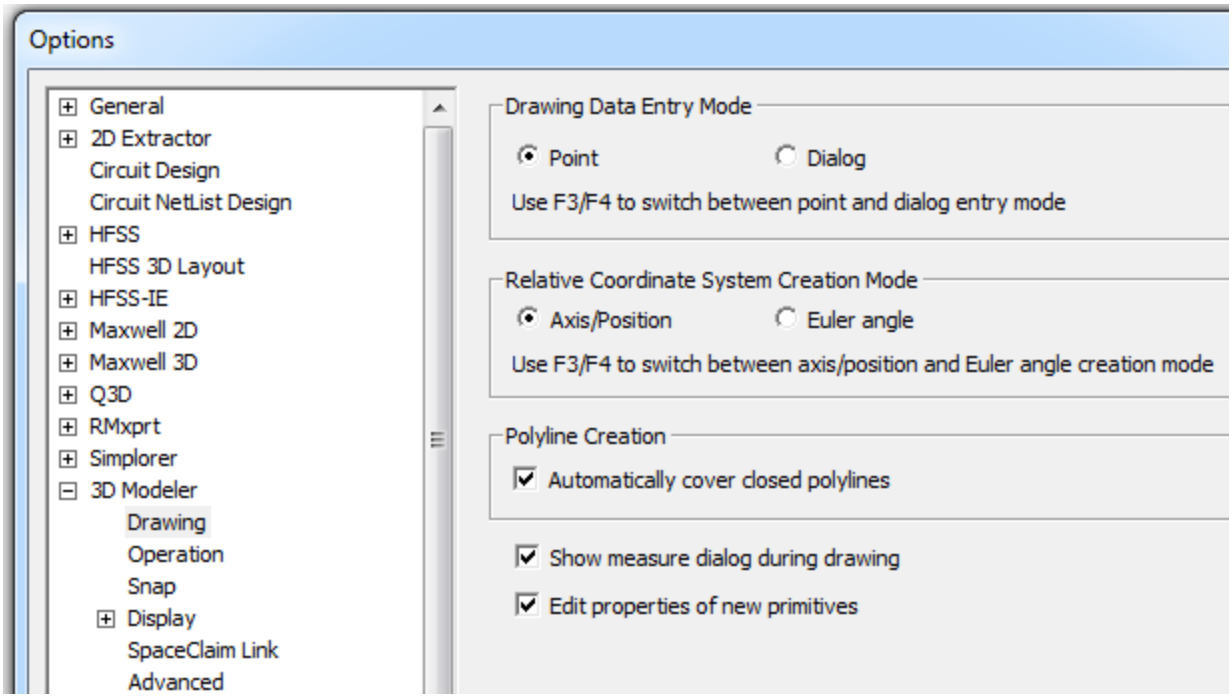


Figure 2-2 A section of the Options window

3. Go to **HFSS>Boundary Assignment** and ensure all check-boxes on this panel are selected.
4. Expand **3D Modeler Options** and select **Drawing**.



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

4. Select the **Automatically cover closed polylines** check box.
5. Select the **Edit properties of new primitives**check box and click **OK.**

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

Insert HFSS design

The icon highlighted in the image below is the **Insert HFSS design** option.

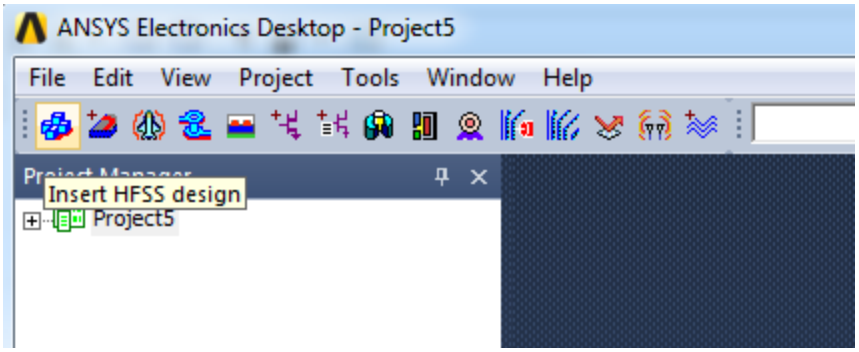


Figure 2-3 Insert HFSS Design

1. Expand the project tree.

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

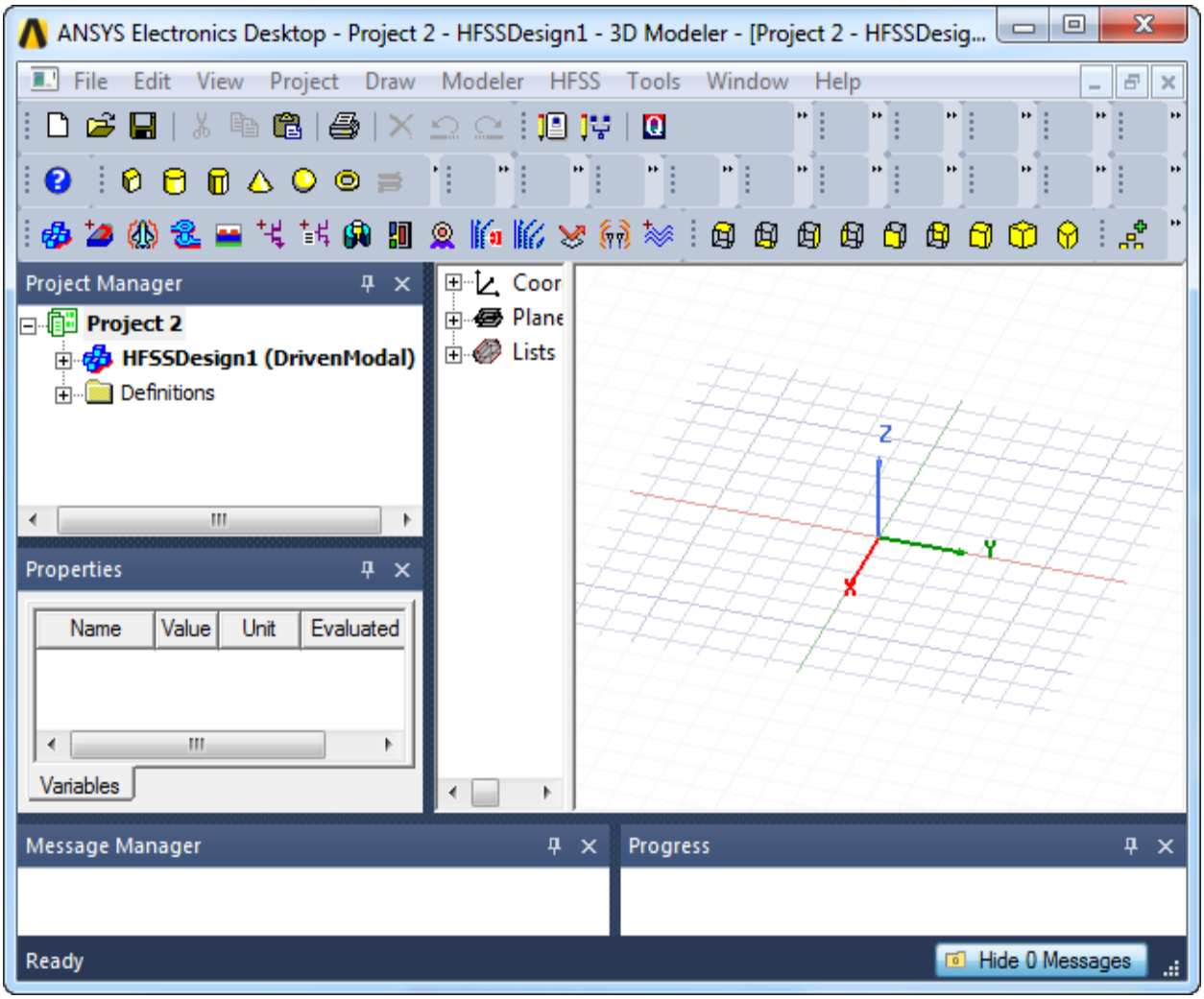


Figure 2-4 Electronics Desktop with the HFSS Design type.

2. Click **Project*n***, hit **F2**, rename the project and save it.

   Note: You can adjust the size of the coordinate system or hide it from **View > Coordinate System**. Similarly, you can adjust grid visibility and its style from **View > Grid Settings**.

Set Model Units (in)

Define the model units as follows:

1. On the toolbar, click **Modeler>Units**.

   The **Set Model Units** 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.

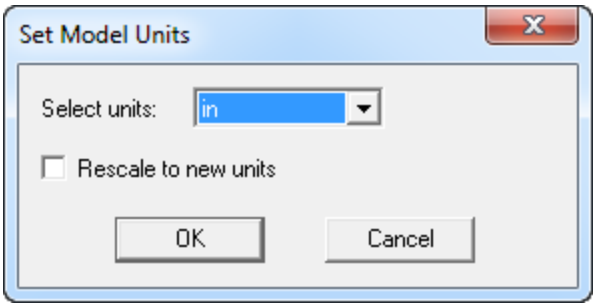


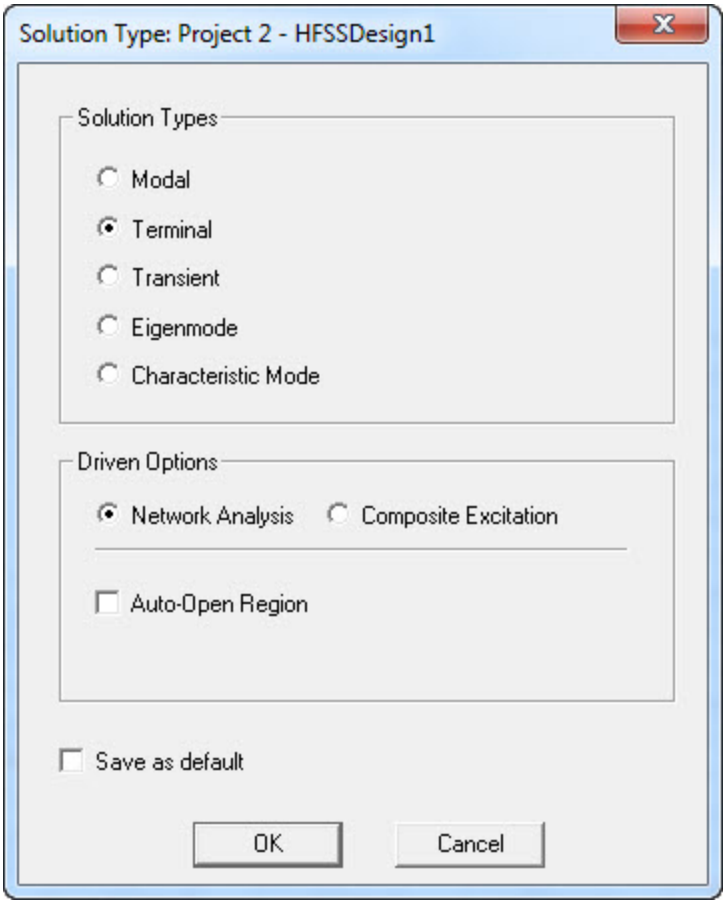
Figure 2-5 Set Model Units dialog box

2. Select **in** (inches) and click **OK**.

Set Solution Type (Terminal)

Specify the design's solution type as follows:

1. On the toolbar, click **HFSS>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.

Figure 2-6 Solution Type dialog box

The **Solution Type** dialog box appears.

2. Select **Terminal**, **Network Analysis** and leave **Auto-Open Region** unchecked, and click **OK**.

   Note Terminal solution type calculates the 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.

3 - Create the 3D Model

This chapter includes the following sections:

- Create the 3-D Model
- Create the air-body
- Create Feed1 (Outer Coax Diameter)
- Create Feed-Pin1 (Coax Inner Diameter)
- Create the Coax Feed Probe1
- Create the Resonators (4 in number)
- Assign Excitation
- Create Rest of the Model by Duplication
- Boundary Display (Optional)

Create the 3-D Model

The 3-D Model of the bandpass filter is made up of multiple geometrical parts. The parts that make up the model are as follows:

- Air Box (the enclosure which contains the filter)
- Feed1 (Coax Outer Diameter)
- Feedpin1 (Coax Inner Diameter)
- Coax Feed Probe1 (Coax Inner Diameter)
- Resonators (4 in number)

For the first two items (and the duplicate of Feed1), assign *vacuum* as the material while for the rest, assign *pec*.

Note Free hand drawing of the objects and then editing properties are easier than creating the objects by entering dimensions, co-ordinates, height, radii, etc. in the status bars in the bottom of the application.

Create the Air Box

To create an air box, draw a box of any size freehand anywhere in the 3D Modeler window as follows.

1. On the menu bar, click **Draw>Box.**

   The cursor changes to a snapping point indicator.

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

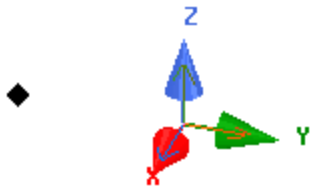


Figure 3-1 Snapping Point indicator

2. Click anywhere in the 3D Modeler window to establish the starting point.
3. Drag your cursor along the **XY** plane, and click a second time to create the base.

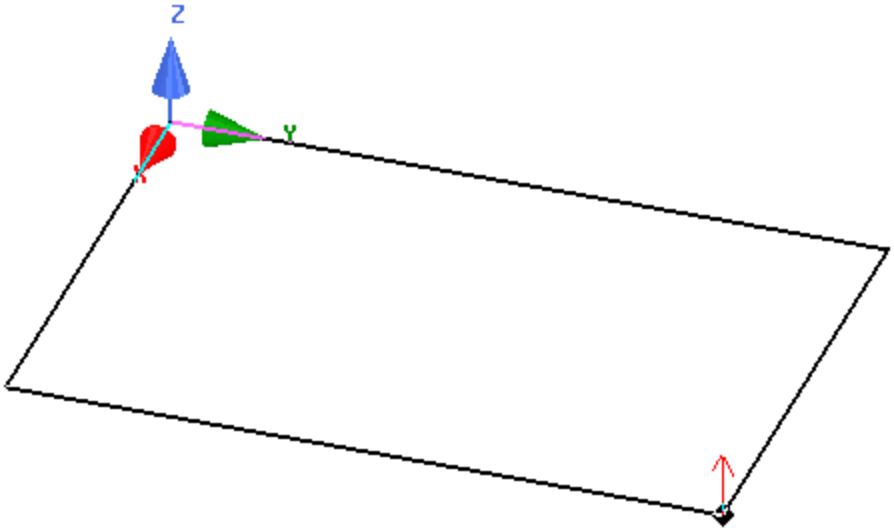


Figure 3-2 The arrow is the mini Z axis.

4. Drag the cursor along the established mini Z axis and then, click, a third time to create the height.

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

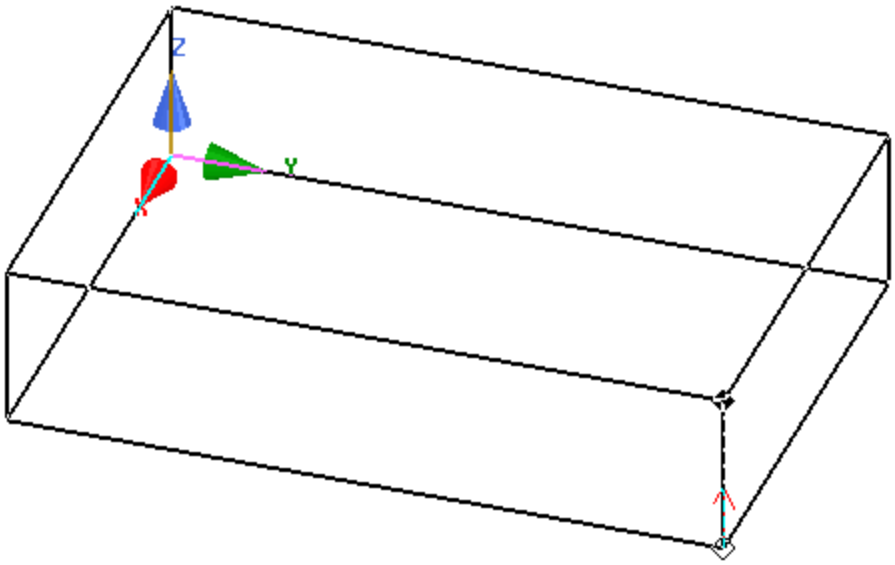


Figure 3-3 Box drawn

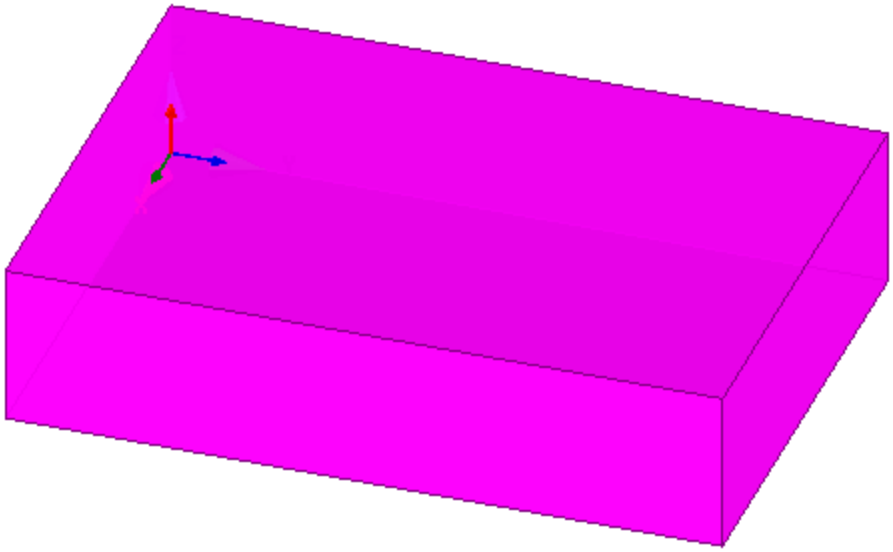


Figure 3-4 Box drawn after the final mouse-click

The **Properties** dialog box appears.

4. On the **Command** tab edit the fields as shown in "Command tab for the first box." on the next page. .

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

5. On the **Attribute** tab rename box to *Air* and select **vacuum** from the **Material** drop-down menu, set the Transparency to 0.8, and click **OK**.
6. Click outside the box and press **CTRL+D** to fit the view.

   Note As you continue to build the model, whenever you want to fit the view press **Ctrl+D**.

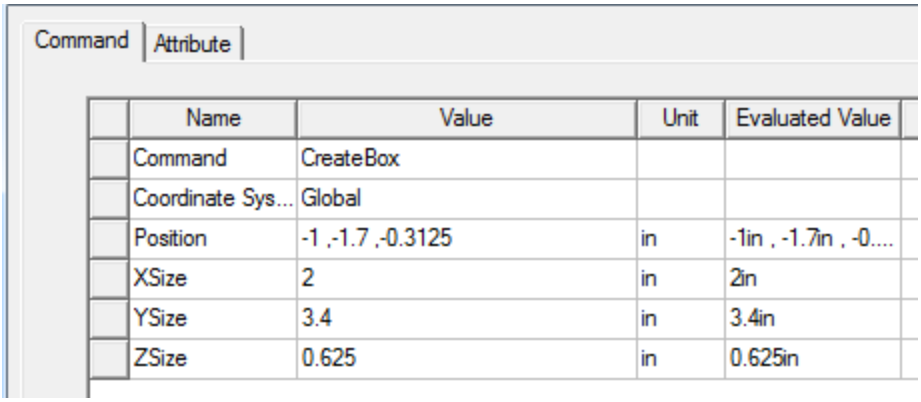
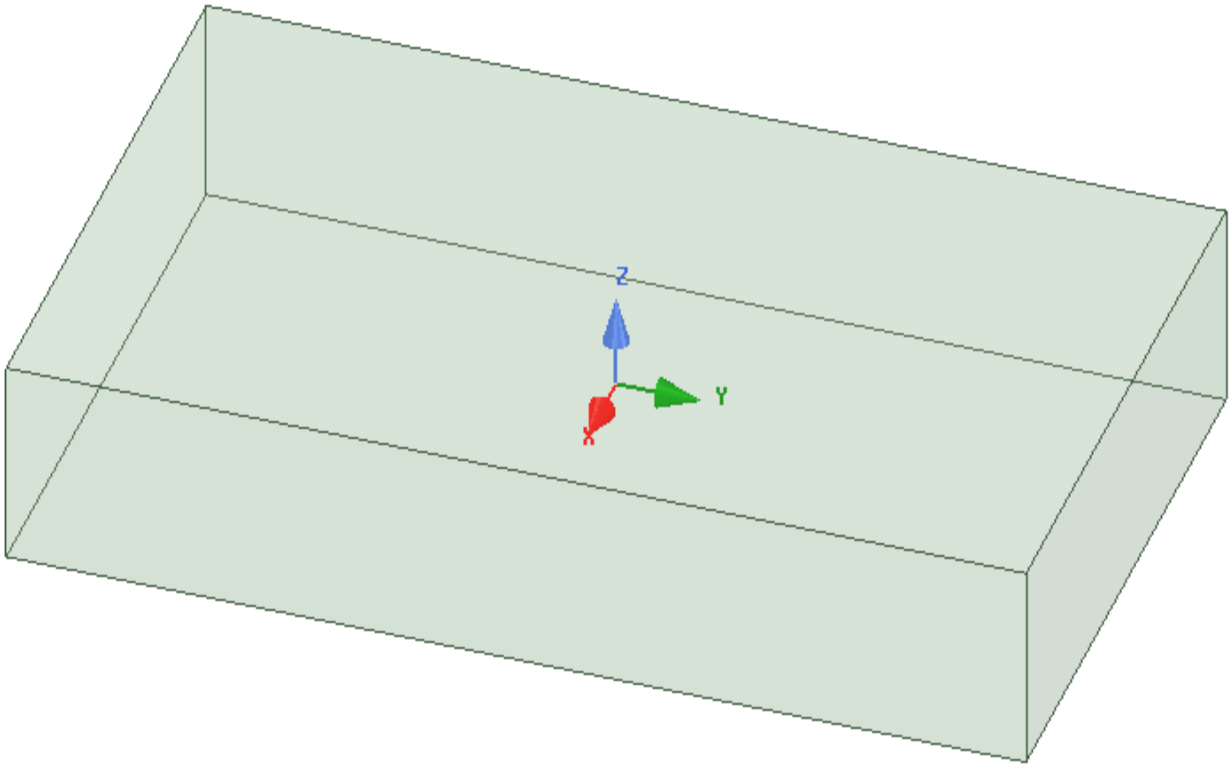


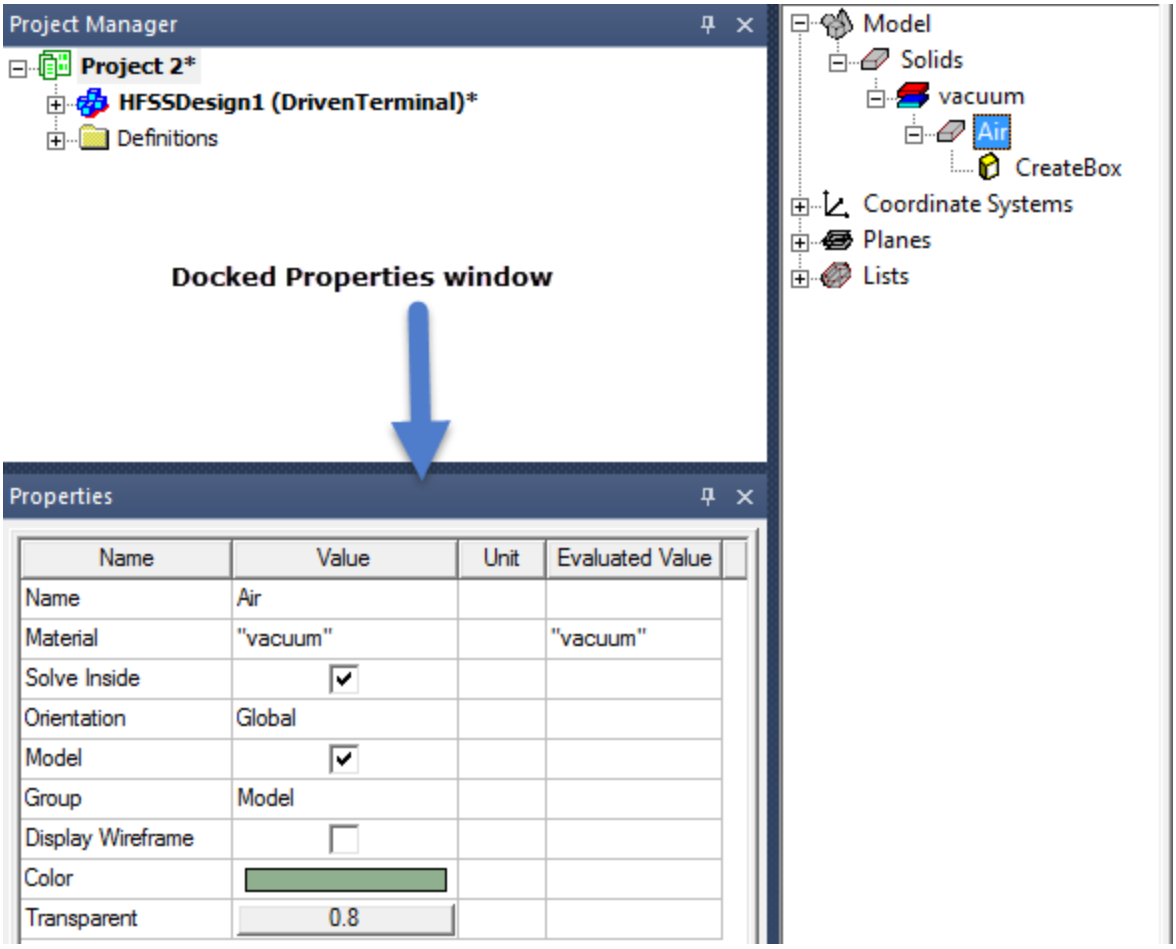
Figure 3-5 Command tab for the first box.



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

Figure 3-6 The box created and placed in the modeler field.

Figure 3-7 When you select a 3D or a 2D object in the history tree, its details can be readily viewed in the docked Properties window. For instance if you select Air in the history tree, its details are displayed in the docked Properties window. You can enable this window from View > Properties.



Create Feed1 (Coax Outer Diameter)

To create Feed1, draw a cylinder freehand as described below.

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

   The active plane changes to YZ.

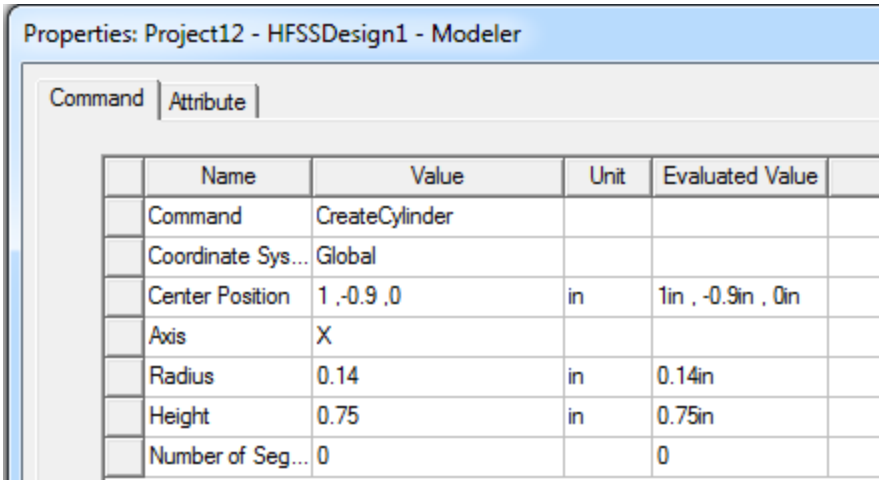
   Note Alternatively, you can simply select **YZ** from the **Drawing Plane** drop-down menu on the HFSS toolbar.

2. Click **Draw>Cylinder** and create the cylinder anywhere in the Modeler window.

   Note In this case, draw the circular cross-section first and then draw the length.

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

3. On the **Command** tab edit the fields as shown in "Command for Feed1" below. and on the **Attribute** tab rename the cylinder to Feed1, set the transparency to 0.4 and click **OK**.



	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	1 ,-0.9 ,0	in	1in , -0.9in , 0in
	Axis	X		
	Radius	0.14	in	0.14in
	Height	0.75	in	0.75in
	Number of Seg...	0		0

Figure 3-8 Command for Feed1

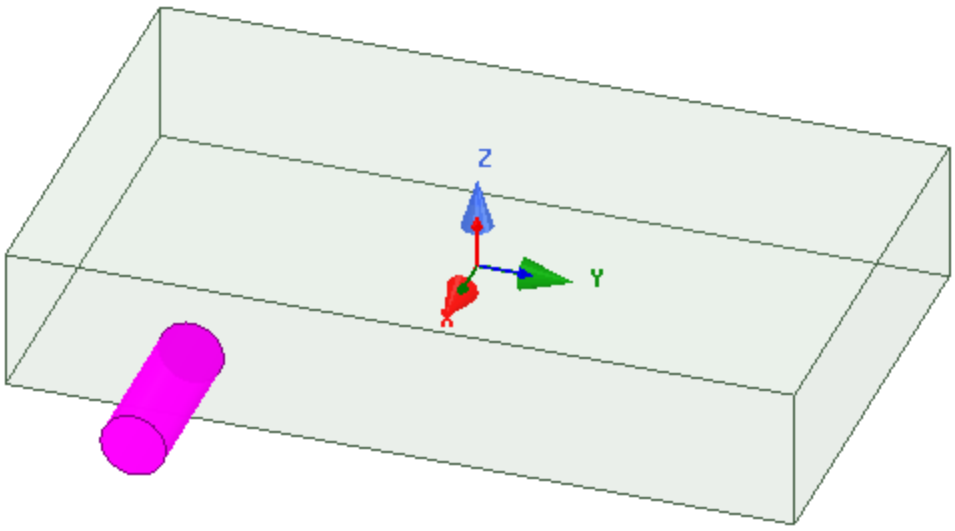


Figure 3-9 Feed1 created.

Create Feed Pin1 (Coax Inner Diameter)

To create the feedpin you will draw a cylinder freehand. You can pick the material from the library before drawing a geometry if it is common for many objects.

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

1. On the Modeler Materials drop-down, choose **Select..**.

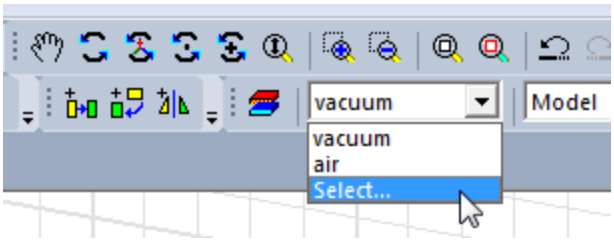


Figure 3-10 toolbar

The **Select Definitions** window appears.

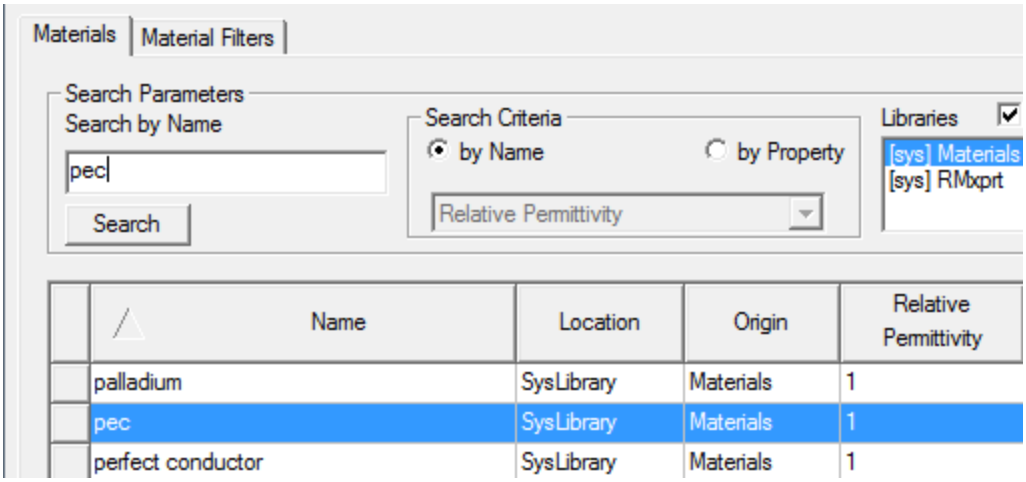


Figure 3-11 Select Definitions window

2. Type *pec* in the **Search by Name** field and press Enter.

   The default material is set to *pec* and the **Select Definition** window closes.
3. Draw a cylinder freehand and edit the fields on the **Command** tab as shown in "feedpin1 Command tab" on the next page. .
4. On the **Attribute** tab rename object to *feedpin1*, set the transparency to 0.4 and 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.

Command | Attribute

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	1 ,-0.9 ,0	in	1in , -0.9in , 0in
	Axis	X		
	Radius	0.06	in	0.06in
	Height	0.75	in	0.75in
	Number of Seg...	0		0

Figure 3-12 feedpin1 Command tab

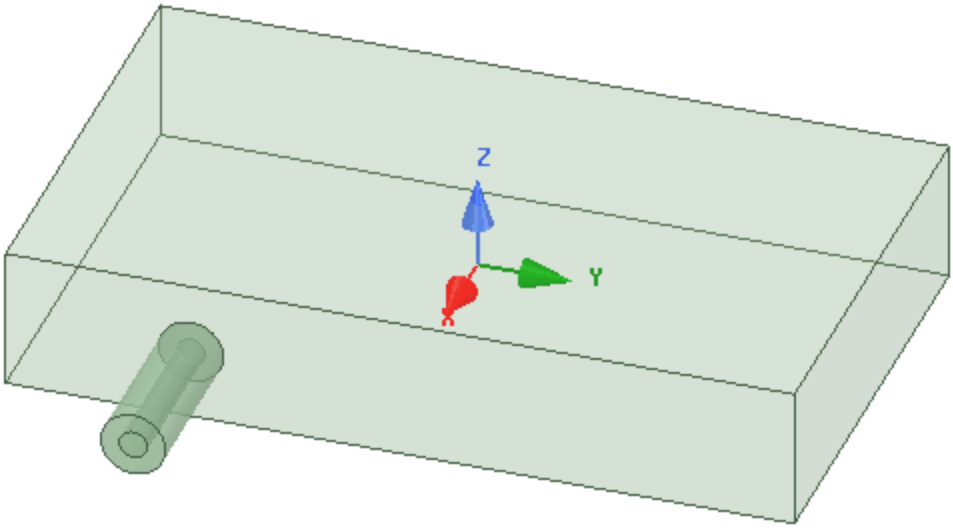


Figure 3-13 The feedpin1 is created.

Create Coax Feed Probe1

1. Draw a cylinder freehand and edit the fields on the Command tab as shown in "Command tab of Coax Feed Probe1" on the facing page.

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 Sys...	Global		
	Center Position	1 ,-0.9 ,0	in	1in , -0.9in , 0in
	Axis	X		
	Radius	0.06	in	0.06in
	Height	-0.15	in	-0.15in
	Number of Seg...	0		0

Figure 3-14 Command tab of Coax Feed Probe1

2. On the Attribute tab rename the object as feedprobe1, set the transparency to 0.4, and click OK.

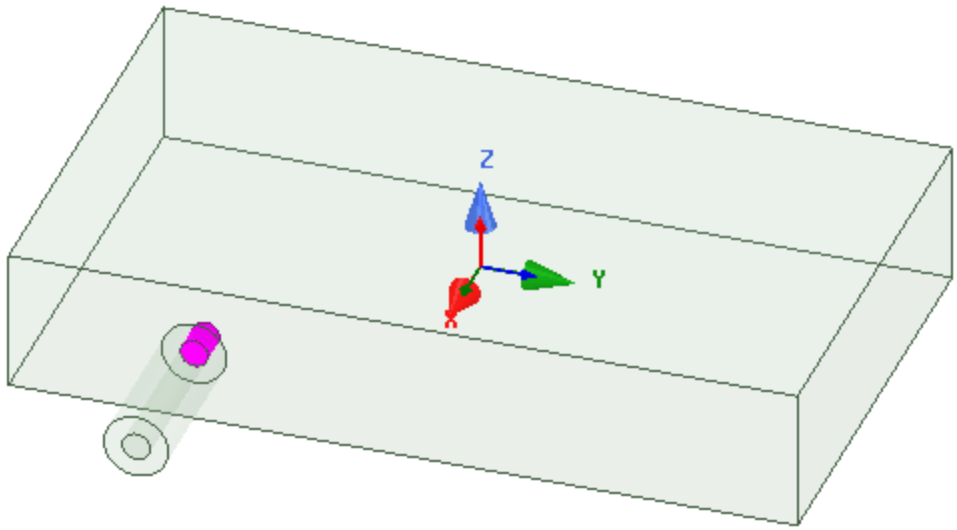


Figure 3-15 Feedprobe1 added.

Create the Resonators

Draw four resonators for this model. For each resonator, draw a box freehand and place it accordingly in the structure drawn until now.

To create I1:

1. Draw a box freehand and on the Command tab edit the fields as shown in "Command tab for I1" on the next page. .

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	CreateBox		
	Coordinate Sys...	Global		
	Position	0.85 ,-0.9625 ,-0.03	in	0.85in , -0.9625in , -0.03in
	XSize	-1.7	in	-1.7in
	YSize	0.125	in	0.125in
	ZSize	0.06	in	0.06in

Figure 3-16 Command tab for I1

2. On the **Attribute** rename the object as *I1* , set the transparency as 0.4 and click **OK**.

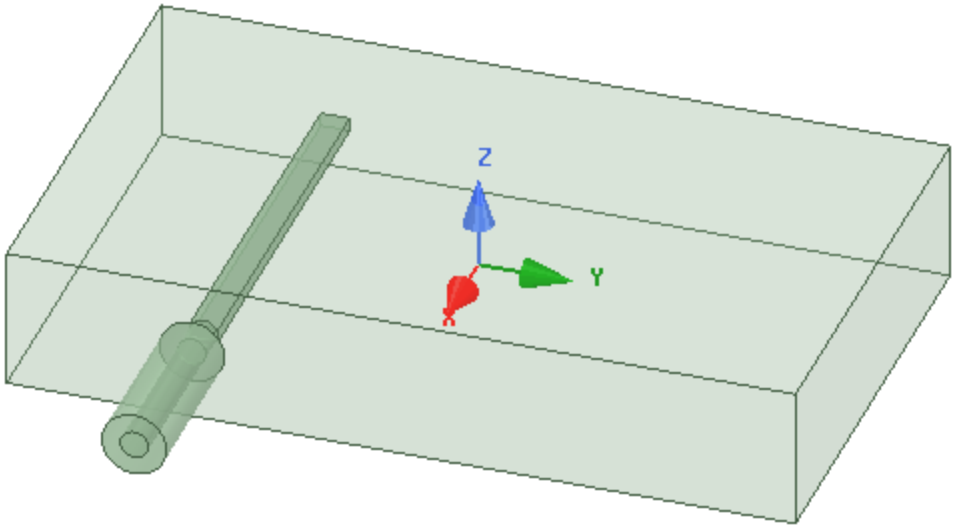


Figure 3-17 The first resonator created

To create I2:

1. Draw a box freehand and set the **Command** dialog box as shown in "Command tab for I2" on the facing page. .

   Note: Alternatively, copy the object I1 in the command history tree, paste it and edit I2 in the **Properties** window,

2. On the **Attribute** tab, rename object as *I2* , set the transparency as 0.4 and 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.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-1 ,-0.75 ,-0.03	in	-1in , -0.75in , -0.03in
	XSize	1.818	in	1.818in
	YSize	0.125	in	0.125in
	ZSize	0.06	in	0.06in

Figure 3-18 Command tab for l2

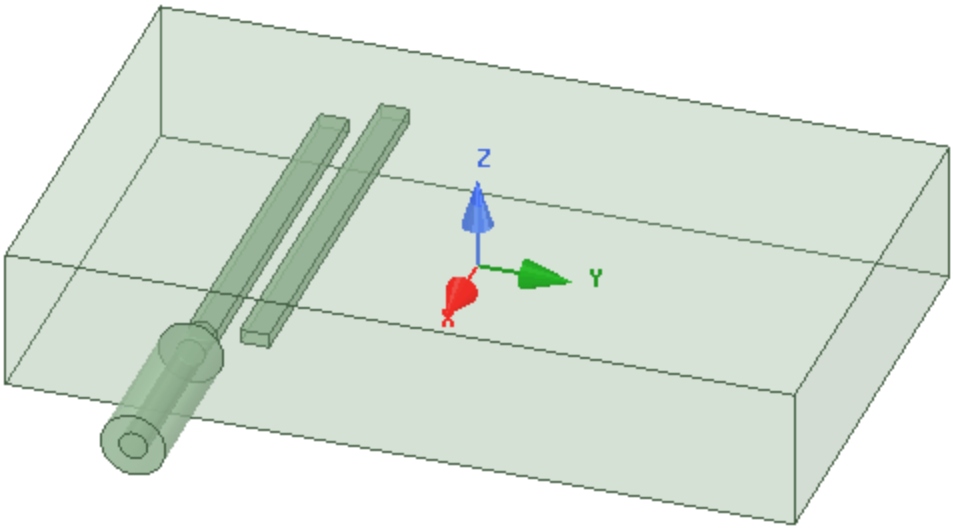


Figure 3-19 The second resonator applied.

To create l3:

1. Draw a box freehand and edit the fields on the **Command** tab as shown in "Command tab for l3 " on the next page. .

   Note: Alternatively, copy the object l2 in the command history tree, paste it and edit l3 in the **Properties** window.

2. On the **Attribute** tab, rename the object as l3, set the transparency as 0.4 and 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.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	1 ,-0.48 ,-0.03	in	1in , -0.48in , -0.03in
	XSize	-1.818	in	-1.818in
	YSize	0.125	in	0.125in
	ZSize	0.06	in	0.06in

Figure 3-20 Command tab for I3

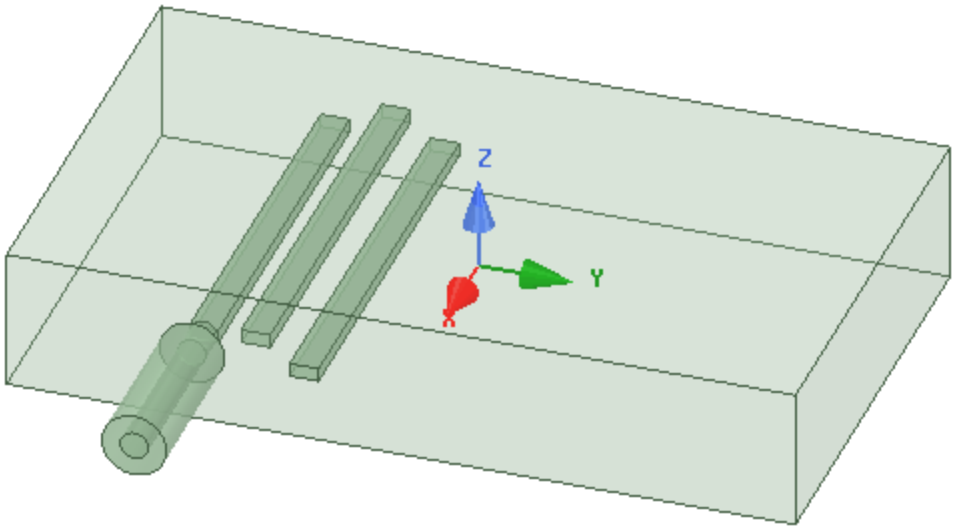


Figure 3-21 The third resonator added

To create I4:

1. Draw a box freehand and set the **Command** dialog box as shown in "Command tab for I4" on the facing page.

   Note: Alternatively, copy the object I3 in the command history tree, paste it and edit I4 in the **Properties** window.

2. On the **Attribute** tab, name it as *I4*, set the transparency to 0.4 and 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.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-1 ,-0.2 ,-0.03	in	-1in , -0.2in , -0.03in
	XSize	1.818	in	1.818in
	YSize	0.125	in	0.125in
	ZSize	0.06	in	0.06in

Figure 3-22 Command tab for I4

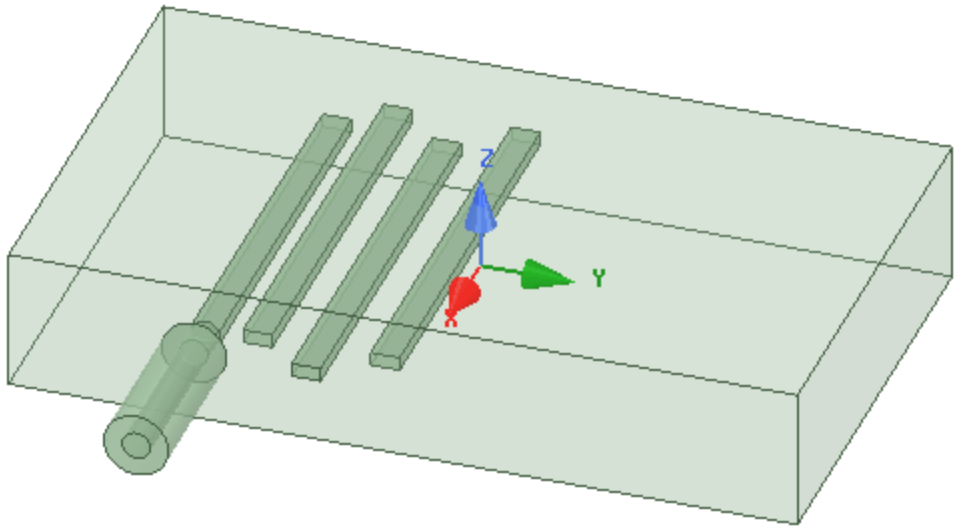


Figure 3-23 The fourth resonator

Assign Excitation

Use wave ports to excite the outer face of the coax connector as described below.

1. To enter face selection mode, either press **F** on your keyboard or click **Edit>Select>Faces**.
2. Select the outer face of the coax line 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.

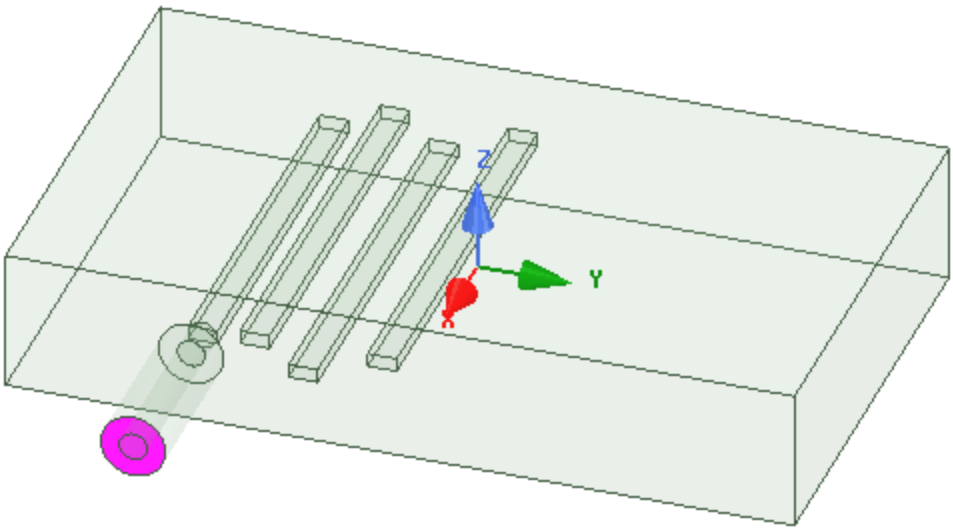


Figure 3-24 Face selected to assign wave port

3. Click **HFSS>Excitations>Assign>Wave Port**

   The **Reference Conductors for Terminals** dialog box appears.

4. Enter a port name and select the option **Use port object name** as shown in "Reference Conductors for Terminals" on the facing page. and 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.

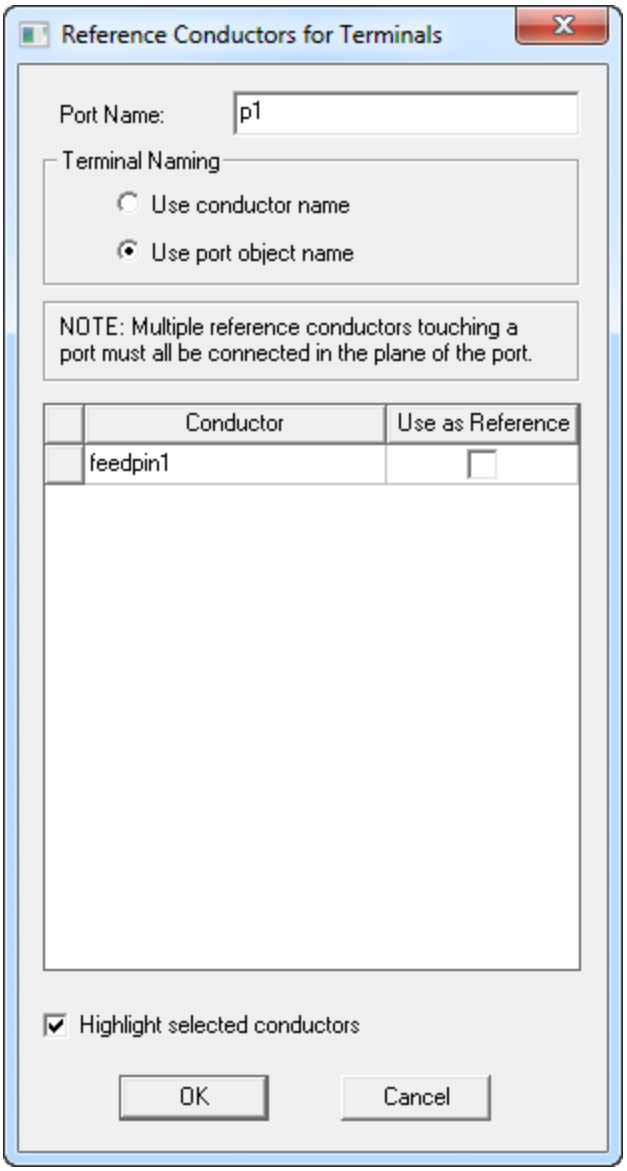


Figure 3-25 Reference Conductors for Terminals

The wave port is assigned and the terminal assignment appears in the **Project** tree under **Excitations**.

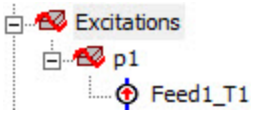
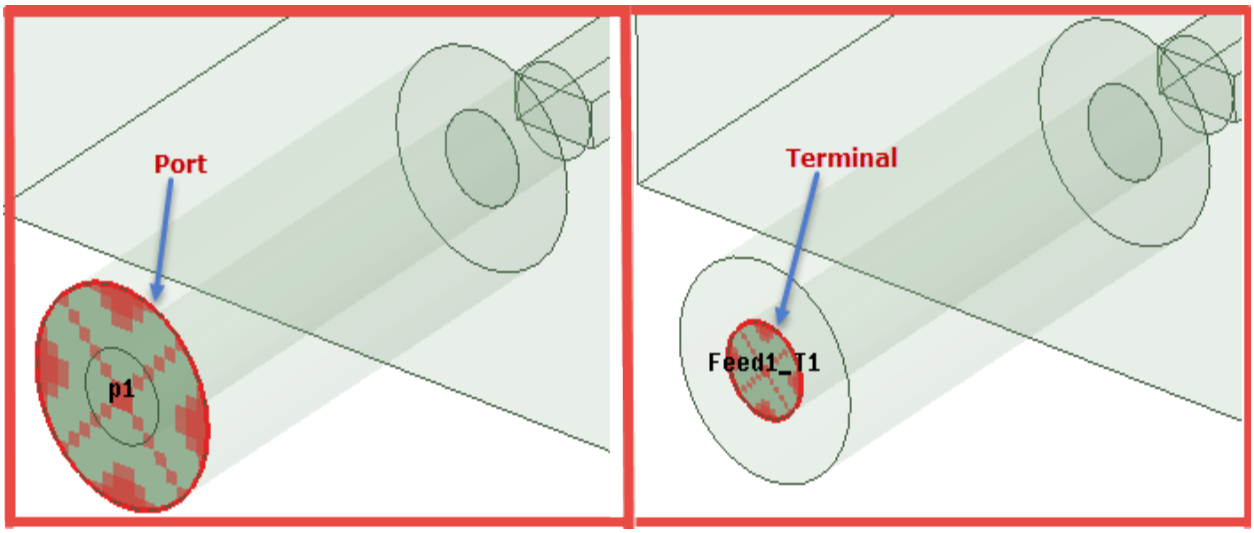


Figure 3-26 Project Tree

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



Create Rest of the Model by Duplication

At this stage, you are almost through with the drawing of your model. All you need to do now is simply duplicate the internal objetcs that you drew as follows.

1. To enter object selection mode, press O on your keyboard or click **Edit>Select>Objects**.
2. Go to **Edit**>**Select**>**By Name**.
   The **Select Object** dialog box appears.
3. Select the following objects and click **OK**: *Feed1*, *feedpin1*, *feedprobe1*, *l1, l2, l3, l4*.

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

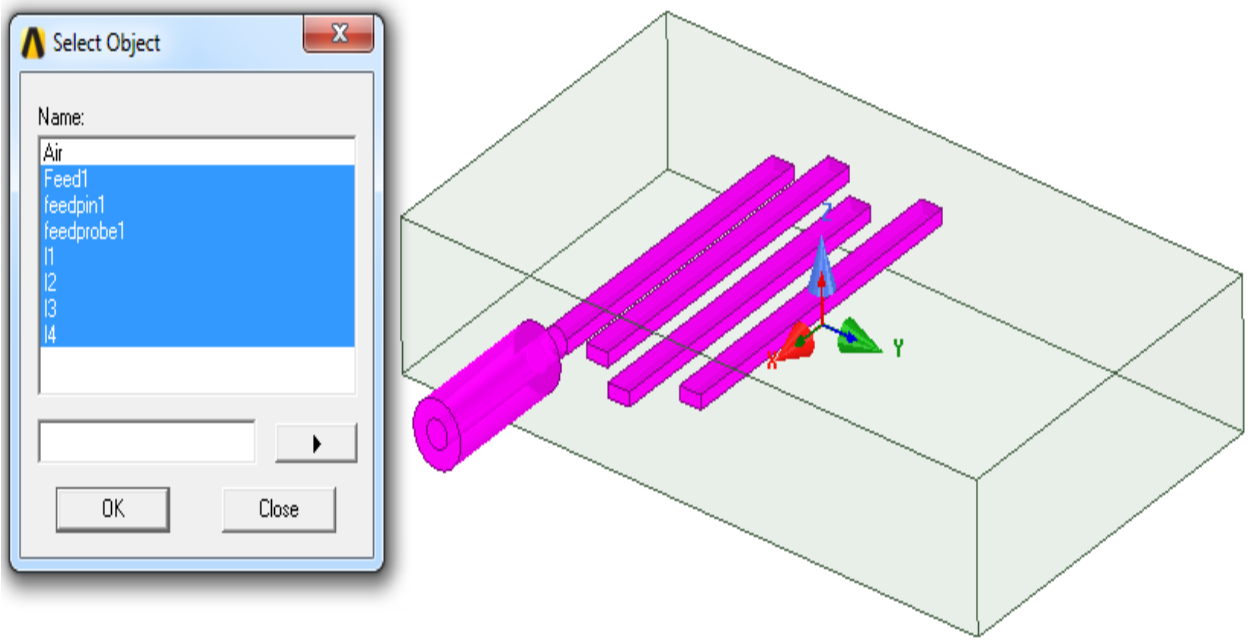


Figure 3-28 Select Object dialog box

4. Click **Edit>Duplicate>Around Axis**

   The **Duplicate Around Axis** dialog box appears.

5. Select **Axis** as Z and specify **Angle** as 180 deg and **Total number** to 2 as shown in "Duplicate Around Axis Z" on the next page. and 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.

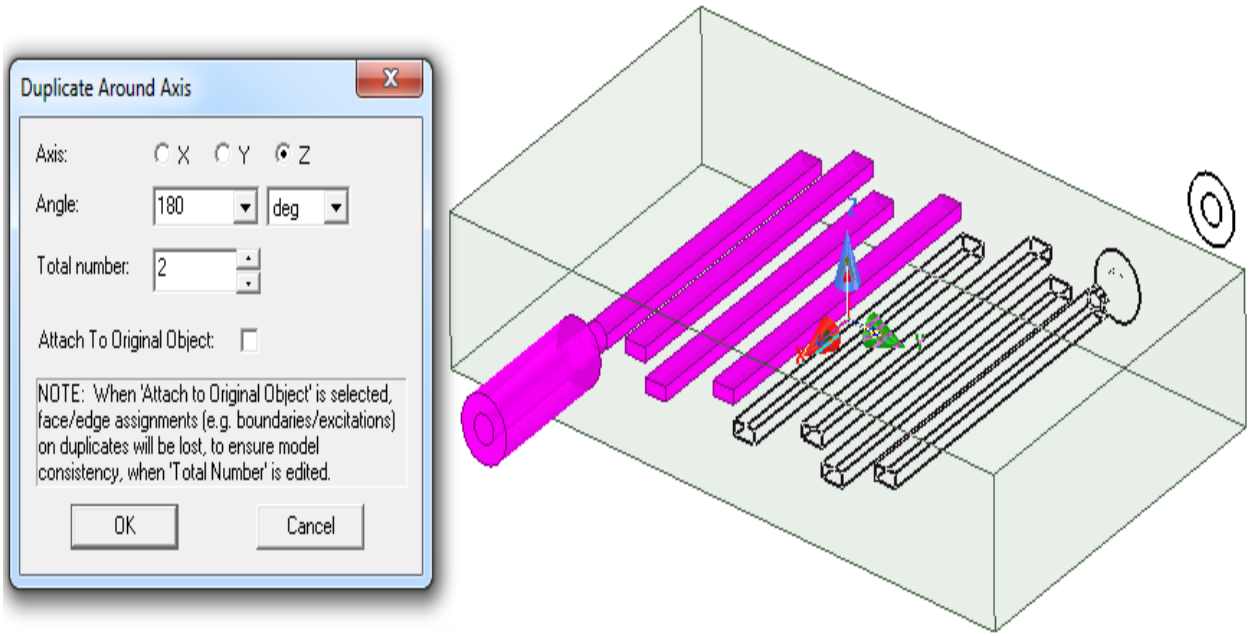


Figure 3-29 Duplicate Around Axis Z

6. Click **OK** to close the **Properties** dialog box that appears.
   The bandpass filter model is updated as shown in "Duplicate of the parts" on the facing page.
   .

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

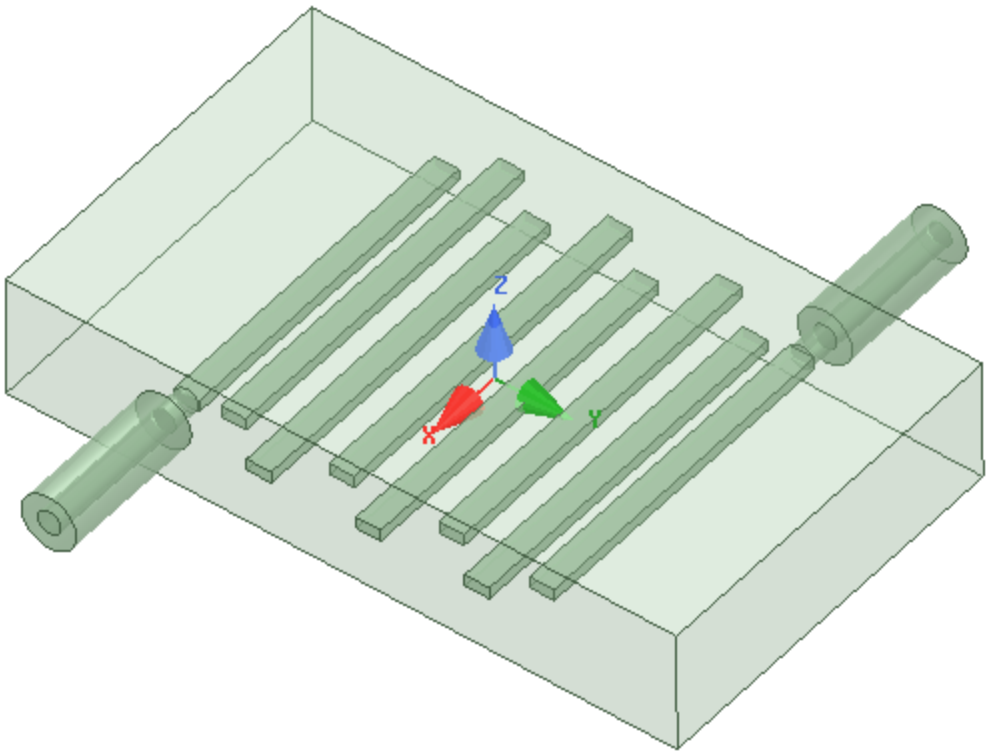


Figure 3-30 Duplicate of the parts

Observe that even the ports and terminals are duplicated and the option **Excitation** in the project tree is updated.

Figure 3-31

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).**

   Note Solver View of Boundaries dialog box appears.

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

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

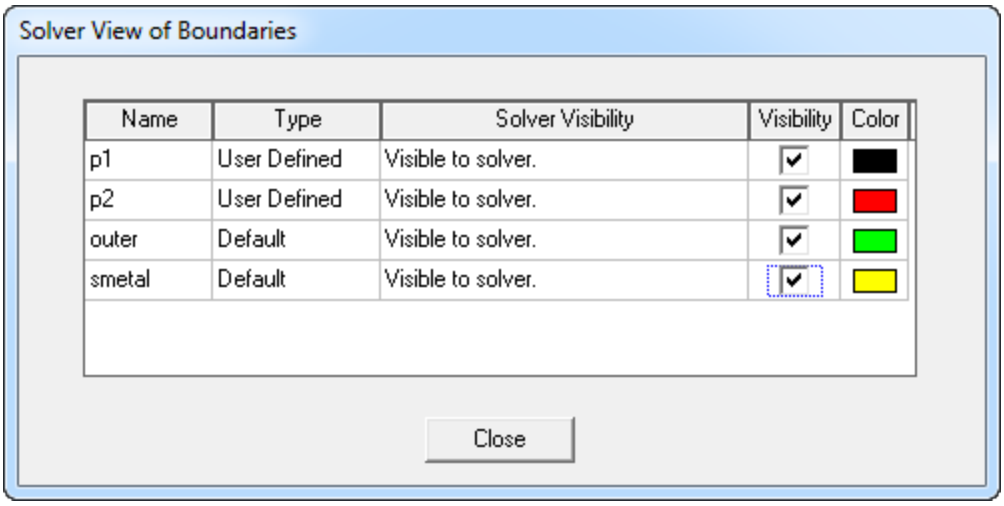


Figure 3-32 Solver View of Boundaries

2. From the **Solver View of Boundaries**, click the **Visibility** check box for the boundaries you wish to display.

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

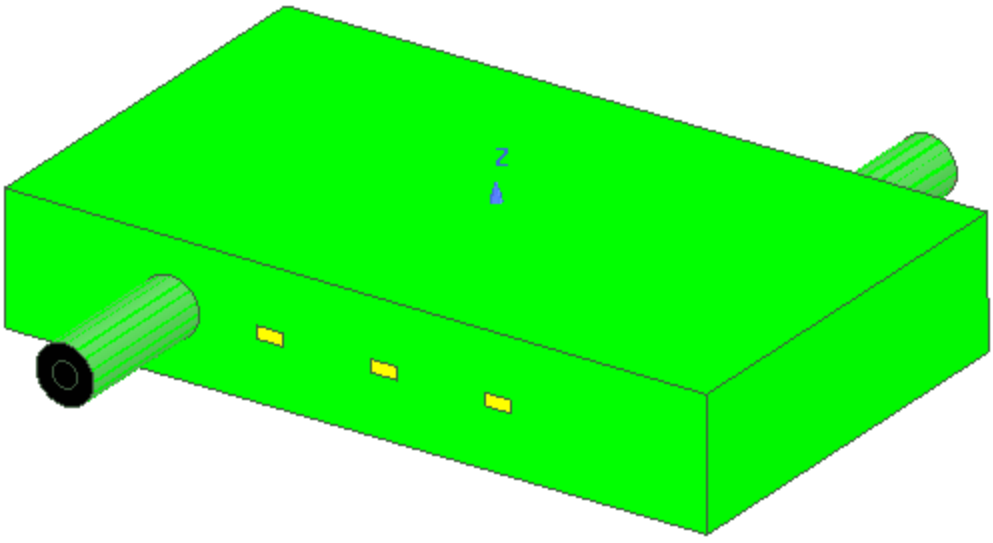


Figure 3-33 Boundaries displayed

3. Click **View>Visibility**>**Active View Visibility**

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

The **Active View Visibility** dialog box appears.

4. Select **Visibility** for the objects you want to see.

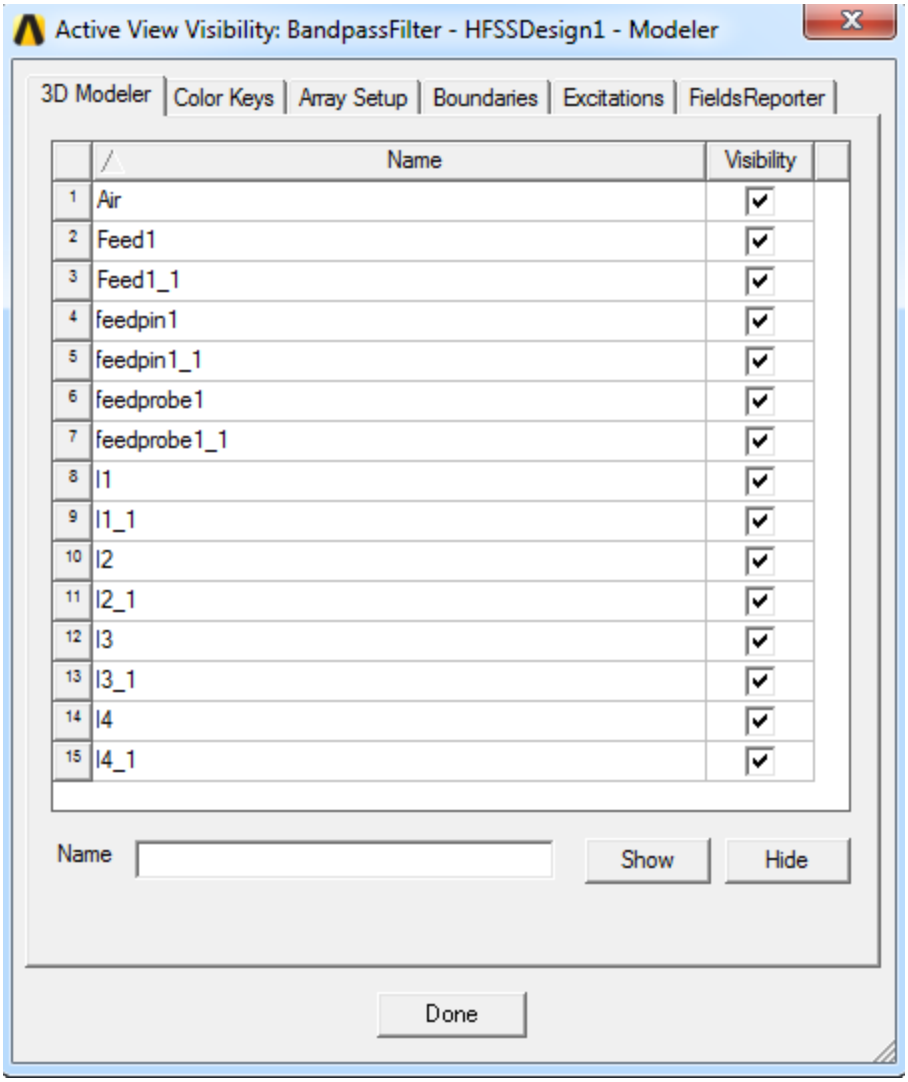


Figure 3-34 Active View Visibility dialog box

Note If you uncheck **Visibility** for all the objects except the ports and boundaries, it makes it easier to see the boundaries. However, remember to check **Visibility** for those objects again to see the whole model.

5. On the **Solver View of Boundaries**, click **Close**.

   On the **Active View Visibility** dialog box, verify that **Visibility** for all the objects is checked and click **Done**.

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.

4 - Analyze the Model

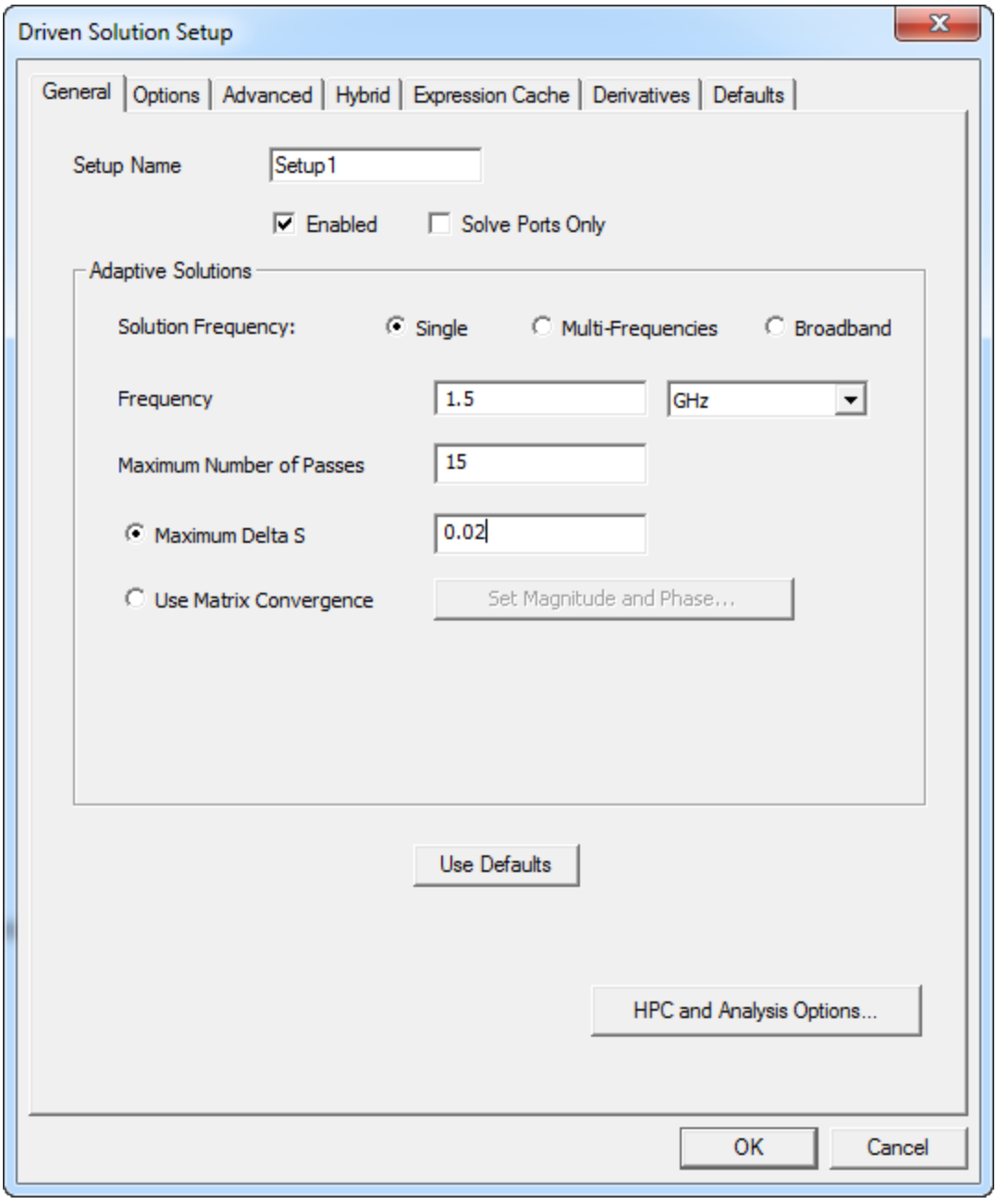
This chapter includes the following sections:

- Add Solution Setup and Frequency Sweep
- Analyze the Bandpass Filter
- Review Solution Data
- Review The Profile Panel
- Review the Convergence Panel
- Review the Matrix Data Panel
- Review the Mesh Statistics Panel
- Create Reports
- Create S Parameter Vs Frequency
- Apply Trace to the Existing Plot
- Change Plot Scale
- Create Field Overlay
- Modify Plot Attributes

Add Solution Setup and Frequency Sweep

1. Right-click **Analysis** on the **Project Manager** and select **Add Solution Setup**
   The **Solution Setup** dialog box appears.
2. Edit the fields on the General tab as shown in "General tab on the Solution Setup dialog box" on the next page. and 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.



General tab on the Solution Setup dialog box

Note This dialog box defines how HFSS automatically generates an accurate mesh and helps you define the stopping criteria for the mesh adaption process.

The Solution setup is created and **Setup1** appears under **Analysis**.

3. Under **Analysis**, right-click **Setup1** and select **Add Frequency Sweep** 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.

The **Edit Frequency Sweep** dialog appears.

4. Edit the fields as shown in "Edit/Frequency Sweep dialog box" below. and click **OK**.

   Note Sweep Type is set to *Fast* because the bandwidth for which we solve this design is narrow.

5. Save the **Project**.

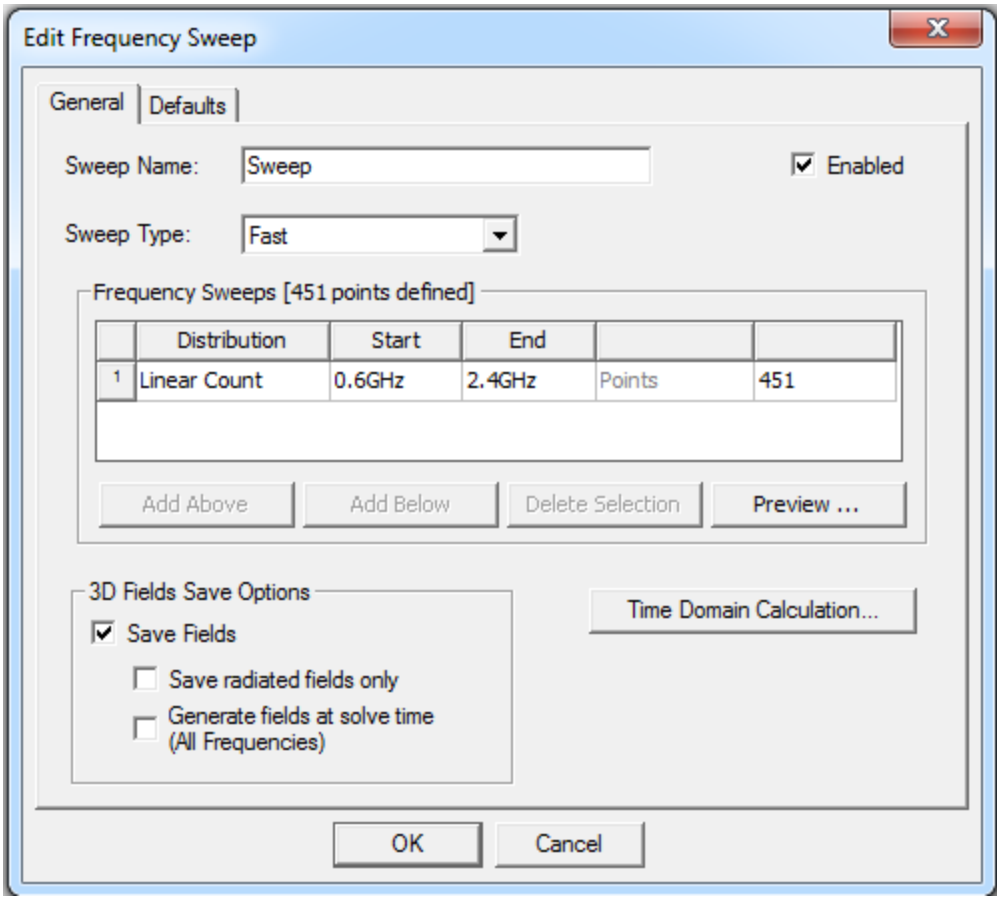


Figure 4-1 Edit/Frequency Sweep dialog box

Add HPC Analysis Setup

HFSS provides the **HPC and Analysis Options** setup for the purpose of high performance computing that ensures efficient simulation. Since you are solving a frequency sweep in this design, using HPC distributes the frequencies across available cores making optimum use of the computational resources.

Note: Ignore this section if you do not want to use HPC.

1. Go to **Tools** > **Options** > **HPC and Analysis Options** 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.

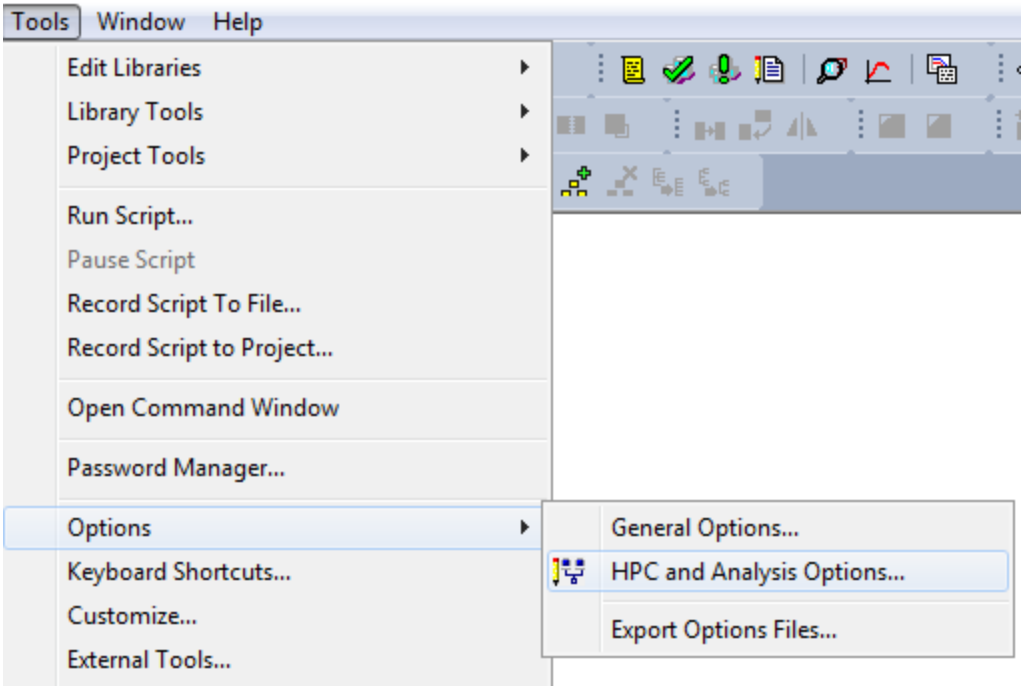
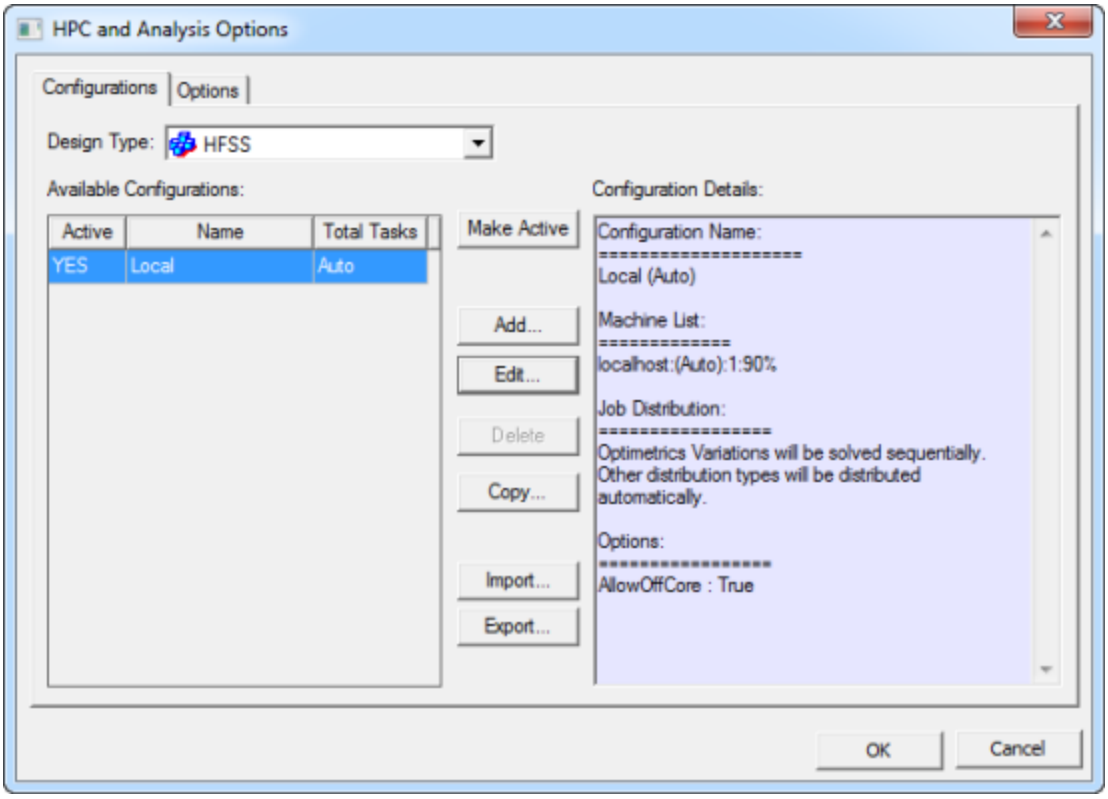


Figure 4-2 Access HPC and Analysis Options

The **HPC and Analysis Options** window 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 HPC and Analysis Options

2. Select the **Design Type** from the drop-down menu for which you want to set up HPC.

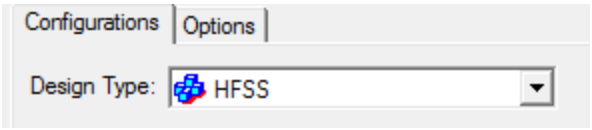


Figure 4-4 Design Type

3. Click the **Add** button.
   The **Analysis Configuration** dialog box appears 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.

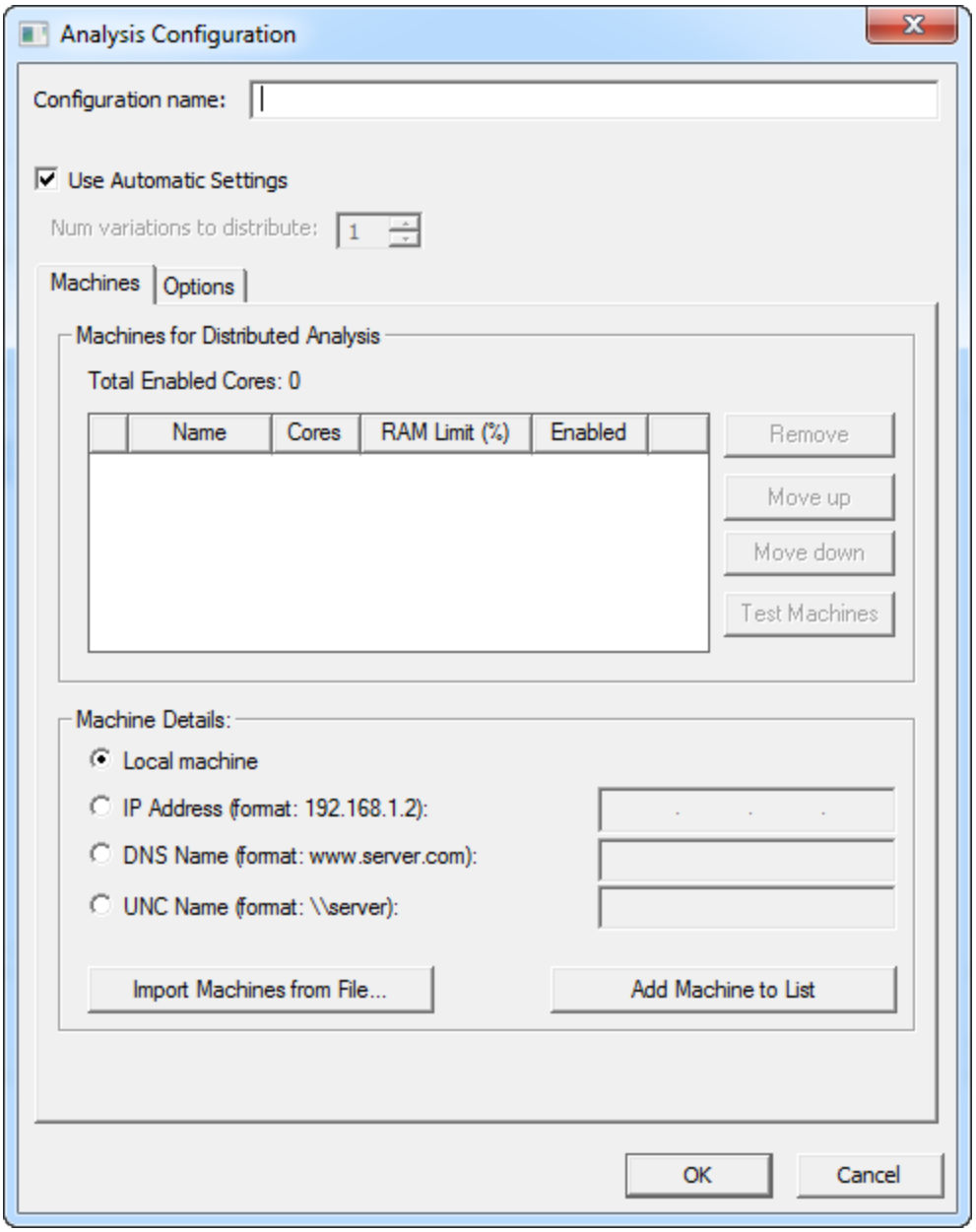


Figure 4-5 Analysis Configuration window

4. Enter a name in the **Configuration name** text box.



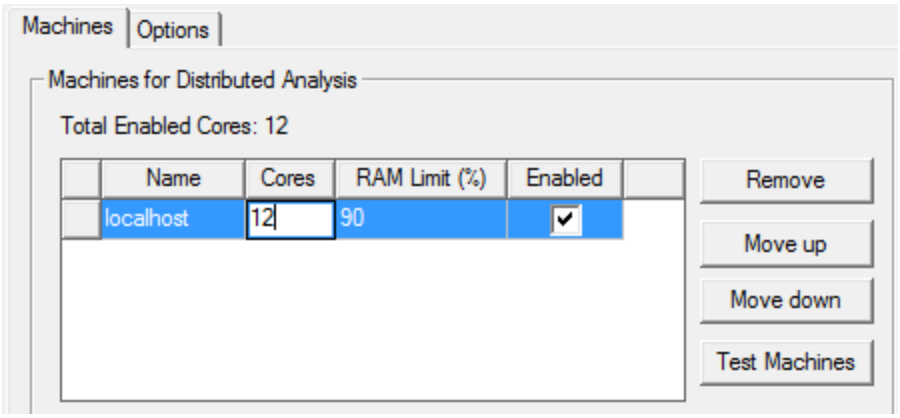
Figure 4-6 Set the Configuration name

5. Select **Use Automatic Settings**.

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

Based on the best use of available resources for the current analysis, this setting automatically selects the number of tasks.

6. On the **Machine Details** panel, select **Local Machine** and click **Add Machine to List**.
7. Set the total number of Cores to 12 as shown below.



Note: The number of tasks is based on a factor associated with the total cores. On a machine with 12 available cores, when the number of tasks is 4, optimum use of the available cores happens. However, if the number of tasks is 5, of the 12 available cores 2 remain idle.

8. Click the **Add Machine to List** button.

   The columns under the **Machines for Distributed Analysis** panel are populated with the total number of enabled tasks and cores.
9. Verify that the settings are as shown in "HPC and Analysis Options" on the next page. and click **OK**.

   Note: Leave the default settings on the Job Distribution and Options tabs as is.

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

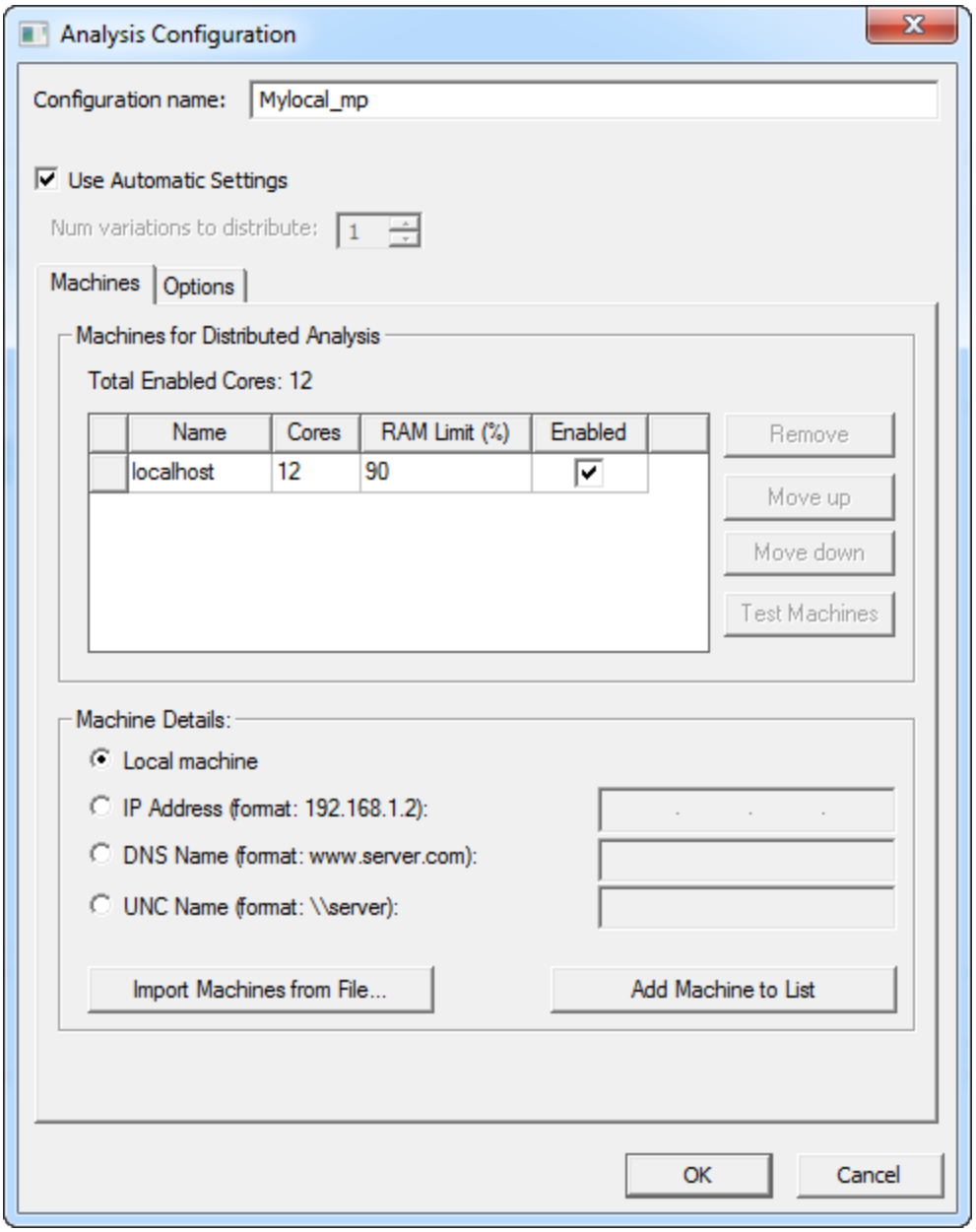


Figure 4-7 HPC and Analysis Options

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

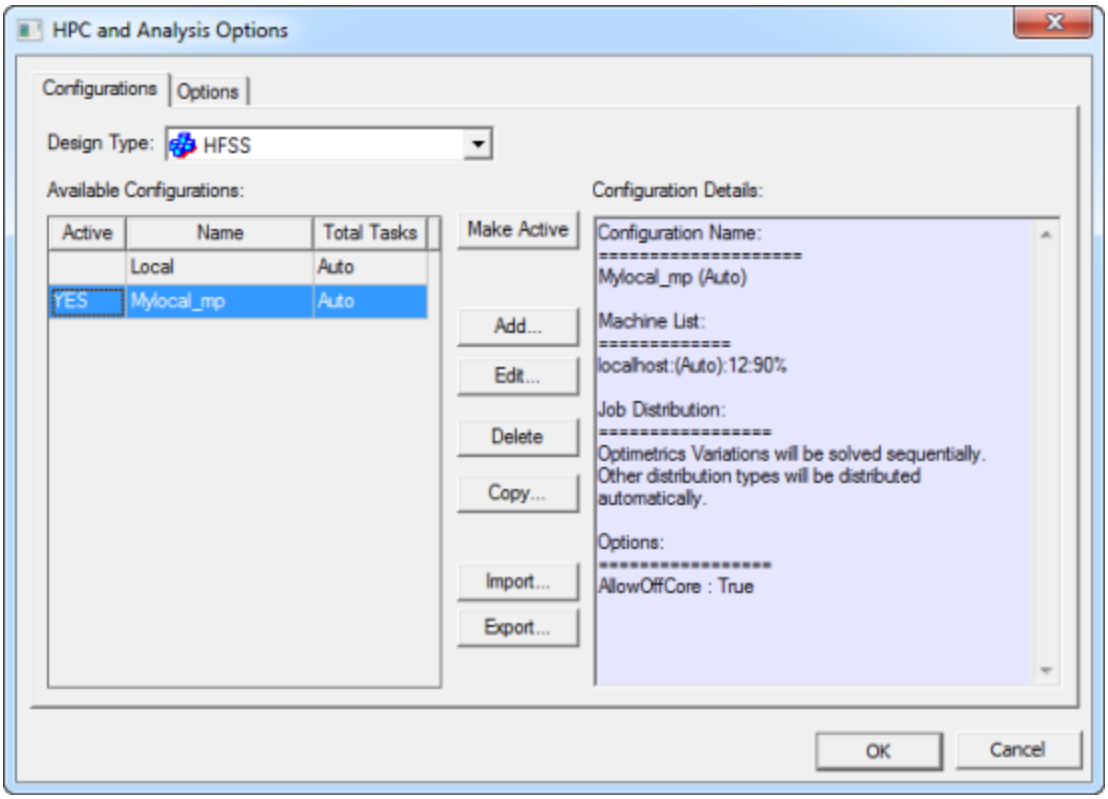


Figure 4-8 HPC and Analysis Options settings

The **Configuration Details** panel and **Available Configurations** panel are updated with the names, number of tasks, and the total cores. HFSS intelligently determines which jobs are to be performed and how to distribute them for the simulation. It automatically apportions the jobs during the simulation process and makes optimum use of the available resources.

For this configuration in the HPC setup, first HFSS uses 12 cores of matrix multiprocessing for the adaptive mesh generation. The number of cores used for solving each frequency point is determined by Total Cores/Number of Tasks. So, after adaptive mesh generation HFSS uses those same 12 cores during the frequency sweep by running 4 frequency points in parallel with 3 cores matrix multiprocessing per frequency point. This solves faster since running frequency points in parallel ensures a more scalabale HPC technology than matrix multiprocessing. However, since 4 frequency points are solved in parallel the analysis uses 4 times the memory of the last adaptive pass. So it is assumed that the machine has adequate memory to solve the 4 frequency points. As an alternative you can use HPC to leverage cores on networked physical nodes which can provide the additional memory for the parallel frequency points.

Analyze the Bandpass Filter

The model has to pass the validation check to confirm your desgin is accurate before you analyze.

1. Click **HFSS>Validation Check**.

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

The **Validation Check** dialog box appears.

Note In HFSS projects, warnings might appear in the **Message Manager** window. Some of these messages are informational warning you of potential problems. It may not always require you to perform any action to deal with the messages.

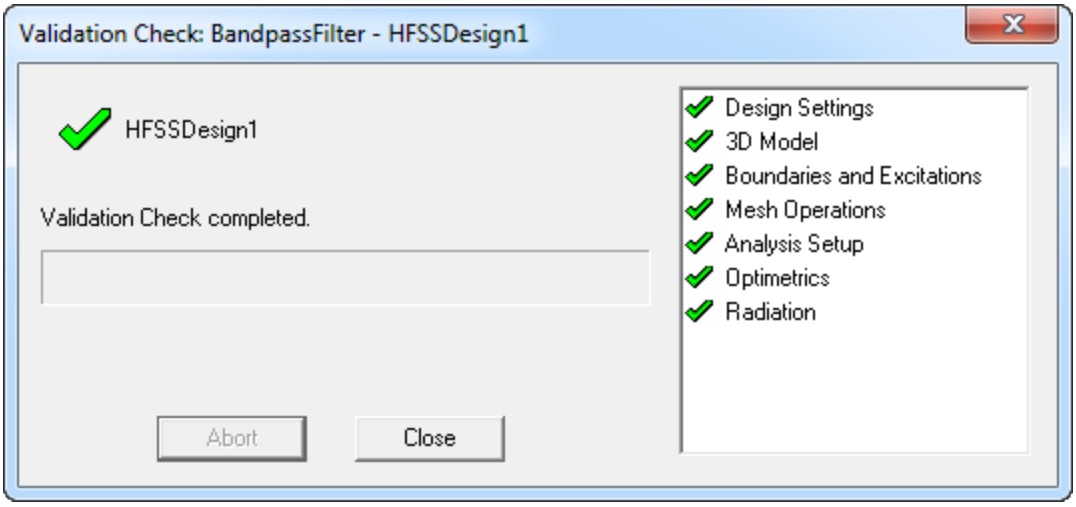


Figure 4-9 Validation Check dialog box.

2. Click **Close**.
3. Click **HFSS>Analyze All.**

Review Solution Data

1. Click **HFSS>Results>Solution Data.**
   The **Solutions** dialog box appears.

   Note Subsequent sections describe all of the panels that constitute the **Solution** dialog box.

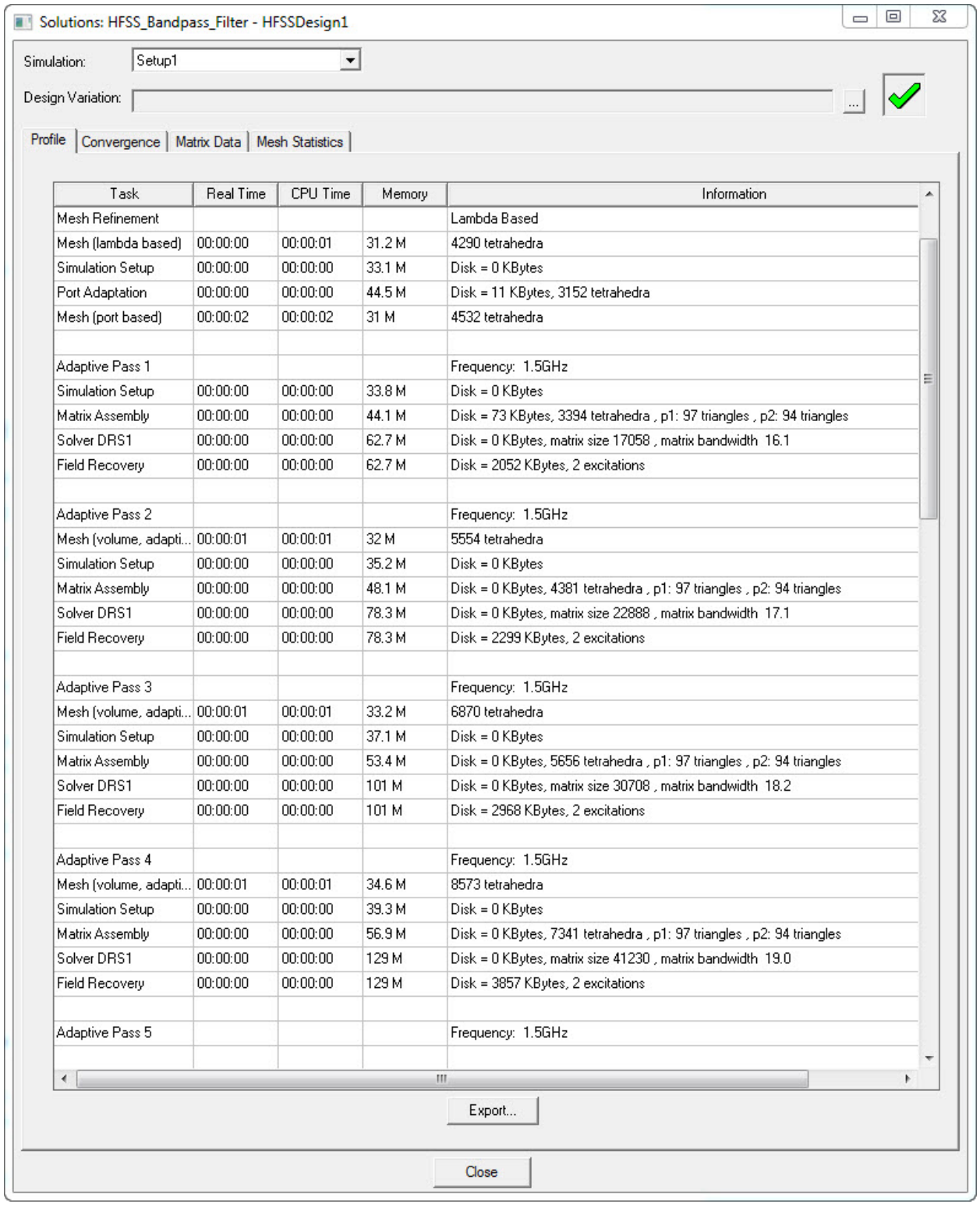
Review the Profile Panel

1. On the **Solutions** dialog box click **Profile**.

The Profile window provides a synopsis of the different stages of the solution process for the extraction of electromagnetic field and SYZ parameter data simulation. The following figure shows the results ranging from mesh generation, the last two different adaptive passes, the matrix assembly and the frequency sweep.

For the Profile window shown in the figure, HPC was enabled. The solution converges in 9 passes. For all the adaptive passes, HFSS employs HPC technology using the 12 cores. After convergence is reached the 4 frequency points are solving in parallel.

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



Task	Real Time	CPU Time	Memory	Information
Mesh Refinement				Lambda Based
Mesh (lambda based)	00:00:00	00:00:01	31.2 M	4290 tetrahedra
Simulation Setup	00:00:00	00:00:00	33.1 M	Disk = 0 KBytes
Port Adaptation	00:00:00	00:00:00	44.5 M	Disk = 11 KBytes, 3152 tetrahedra
Mesh (port based)	00:00:02	00:00:02	31 M	4532 tetrahedra
Adaptive Pass 1				Frequency: 1.5GHz
Simulation Setup	00:00:00	00:00:00	33.8 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	44.1 M	Disk = 73 KBytes, 3394 tetrahedra , p1: 97 triangles , p2: 94 triangles
Solver DRS1	00:00:00	00:00:00	62.7 M	Disk = 0 KBytes, matrix size 17058 , matrix bandwidth 16.1
Field Recovery	00:00:00	00:00:00	62.7 M	Disk = 2052 KBytes, 2 excitations
Adaptive Pass 2				Frequency: 1.5GHz
Mesh (volume, adapti...	00:00:01	00:00:01	32 M	5554 tetrahedra
Simulation Setup	00:00:00	00:00:00	35.2 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	48.1 M	Disk = 0 KBytes, 4381 tetrahedra , p1: 97 triangles , p2: 94 triangles
Solver DRS1	00:00:00	00:00:00	78.3 M	Disk = 0 KBytes, matrix size 22888 , matrix bandwidth 17.1
Field Recovery	00:00:00	00:00:00	78.3 M	Disk = 2299 KBytes, 2 excitations
Adaptive Pass 3				Frequency: 1.5GHz
Mesh (volume, adapti...	00:00:01	00:00:01	33.2 M	6870 tetrahedra
Simulation Setup	00:00:00	00:00:00	37.1 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	53.4 M	Disk = 0 KBytes, 5656 tetrahedra , p1: 97 triangles , p2: 94 triangles
Solver DRS1	00:00:00	00:00:00	101 M	Disk = 0 KBytes, matrix size 30708 , matrix bandwidth 18.2
Field Recovery	00:00:00	00:00:00	101 M	Disk = 2968 KBytes, 2 excitations
Adaptive Pass 4				Frequency: 1.5GHz
Mesh (volume, adapti...	00:00:01	00:00:01	34.6 M	8573 tetrahedra
Simulation Setup	00:00:00	00:00:00	39.3 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	56.9 M	Disk = 0 KBytes, 7341 tetrahedra , p1: 97 triangles , p2: 94 triangles
Solver DRS1	00:00:00	00:00:00	129 M	Disk = 0 KBytes, matrix size 41230 , matrix bandwidth 19.0
Field Recovery	00:00:00	00:00:00	129 M	Disk = 3857 KBytes, 2 excitations
Adaptive Pass 5				Frequency: 1.5GHz

Figure 4-10 Profile with the first few adaptive passes

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

Review the Convergence Panel

1. On the **Solutions** dialog box, click **Convergence**.

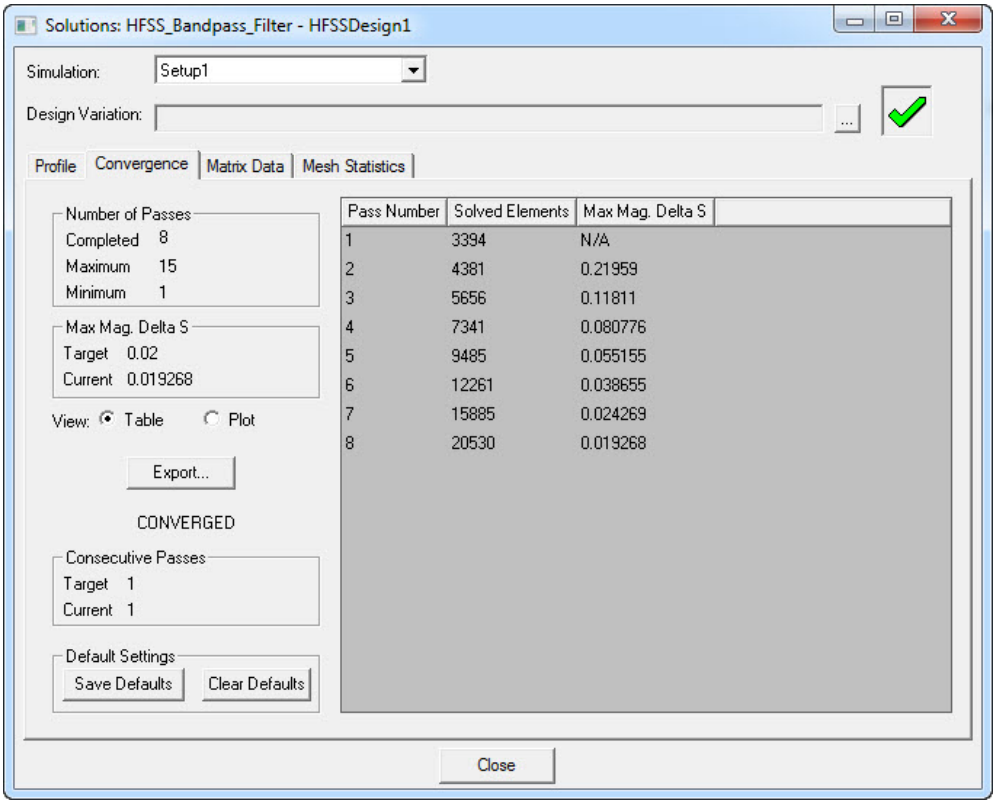


Figure 4-11 Convergence Data Table

The table showing **Convergence** data is displayed.

2. Click the option button **Plot**.

   A graph of the convergence data is displayed.

3. Select the **X** and **Y** axes for your plot from the drop-down menu as shown in "X, Y drop down menus" below. .

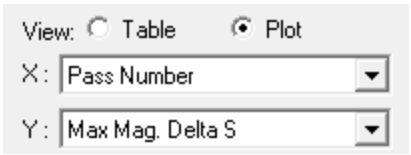


Figure 4-12 X, Y drop down menus

The plot for *Max Mag. Delta S* and *Pass Number* appears. See Figure.

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

Note Other graphs are displayed as you change the X, Y fields from the drop-down menus.

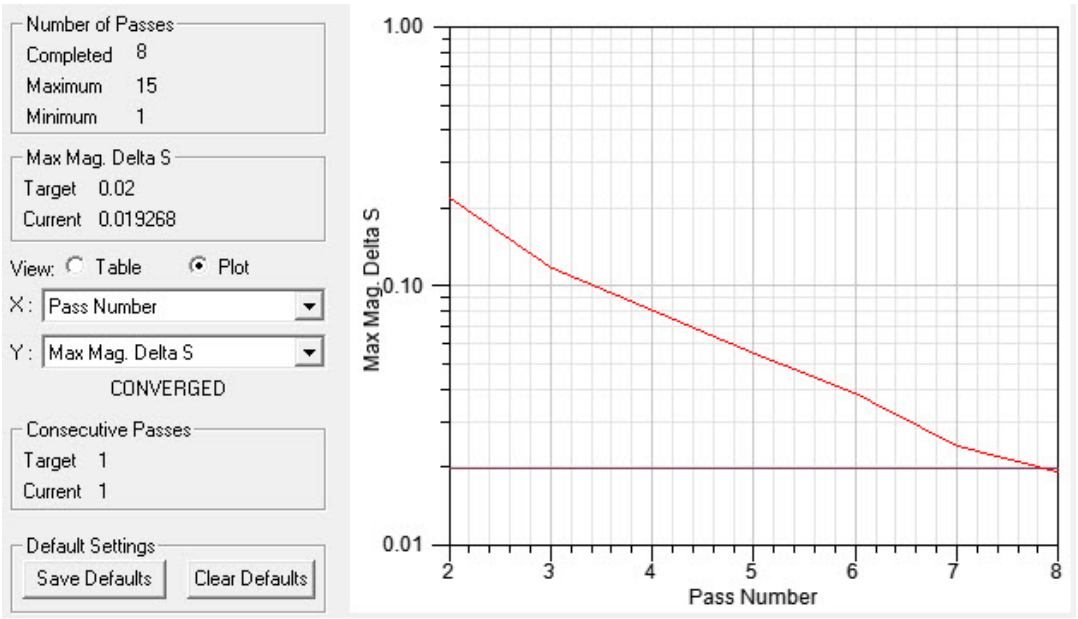
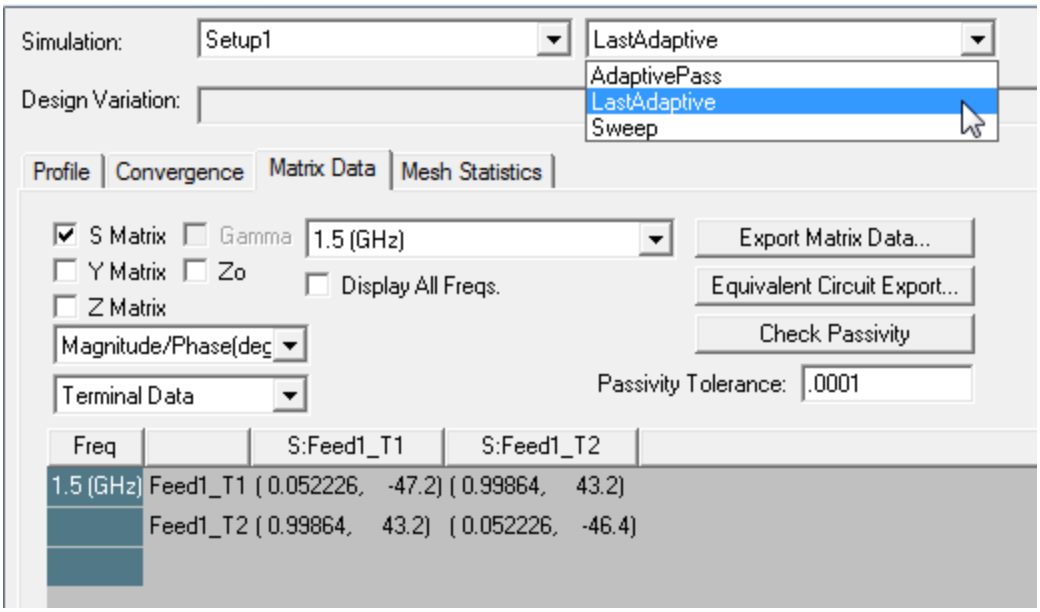


Figure 4-13 The Convergence Graph

Review Matrix Data Panel

Click **Matrix Data** to view information about the S-parameters. For a real-time update of the Matrix Data, set the **Simulation** to *Setup1*, *Last Adaptive*.

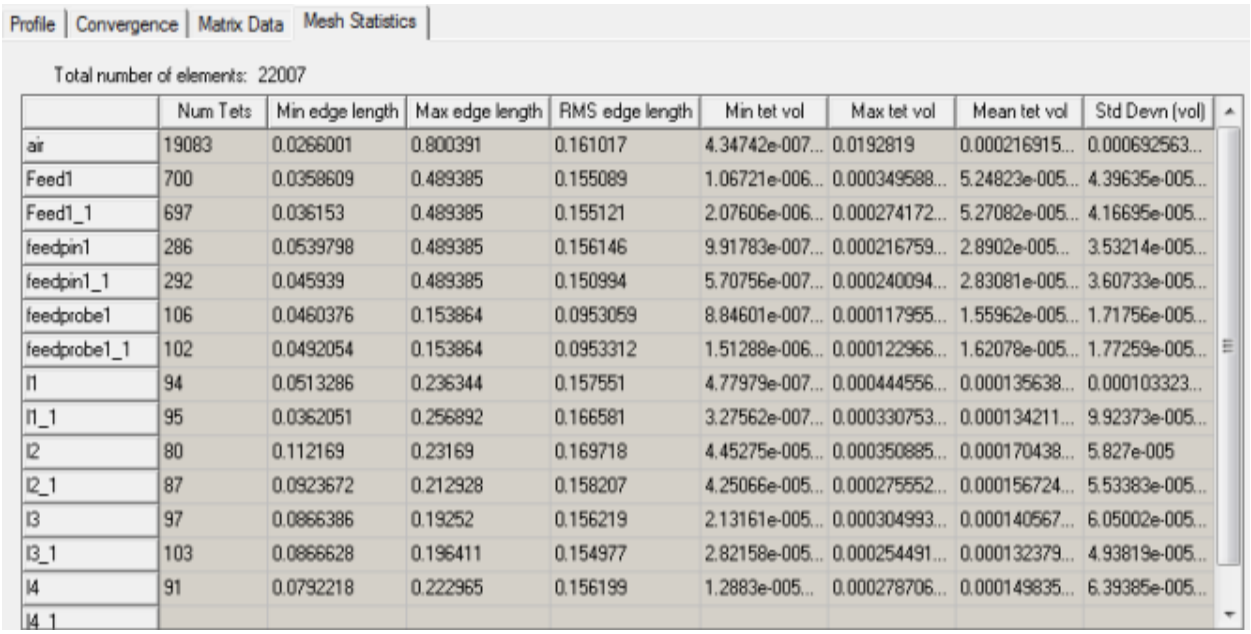


ANSYS Electromagnetics Suite 18.1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4-14 Matrix Data panel

Review Mesh Statistics Panel

1. Click **Mesh Statistics** to see the information about the tetrahedra that were solved for the individual components.



	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
air	19083	0.0266001	0.800391	0.161017	4.34742e-007...	0.0192819	0.000216915...	0.000692563...
Feed1	700	0.0358609	0.489385	0.155089	1.06721e-006...	0.000349588...	5.24823e-005...	4.39635e-005...
Feed1_1	697	0.036153	0.489385	0.155121	2.07606e-006...	0.000274172...	5.27082e-005...	4.16695e-005...
feedpin1	286	0.0539798	0.489385	0.156146	9.91783e-007...	0.000216759...	2.8902e-005...	3.53214e-005...
feedpin1_1	292	0.045939	0.489385	0.150994	5.70756e-007...	0.000240094...	2.83081e-005...	3.60733e-005...
feedprobe1	106	0.0460376	0.153864	0.0953059	8.84601e-007...	0.000117955...	1.55962e-005...	1.71756e-005...
feedprobe1_1	102	0.0492054	0.153864	0.0953312	1.51288e-006...	0.000122966...	1.62078e-005...	1.77259e-005...
l1	94	0.0513286	0.236344	0.157551	4.77979e-007...	0.000444556...	0.000135638...	0.000103323...
l1_1	95	0.0362051	0.256892	0.166581	3.27562e-007...	0.000330753...	0.000134211...	9.92373e-005...
l2	80	0.112169	0.23169	0.169718	4.45275e-005...	0.000350885...	0.000170438...	5.827e-005
l2_1	87	0.0923672	0.212928	0.158207	4.25066e-005...	0.000275552...	0.000156724...	5.53383e-005...
l3	97	0.0866386	0.19252	0.156219	2.13161e-005...	0.000304993...	0.000140567...	6.05002e-005...
l3_1	103	0.0866628	0.196411	0.154977	2.82158e-005...	0.000254491...	0.000132379...	4.93819e-005...
l4	91	0.0792218	0.222965	0.156199	1.2883e-005...	0.000278706...	0.000149835...	6.39385e-005...
l4_1								

Figure 4-15 Mesh Statistics

Create Reports

You can generate reports to further study the simulation results.

Create S-Parameter vs. Frequency

1. Click **HFSS>Results>Create Terminal Solution Data Report>Rectangular Plot**
   The **Report** dialog box appears.
2. Set the fields as in "Report dialog box" on the facing page. .

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

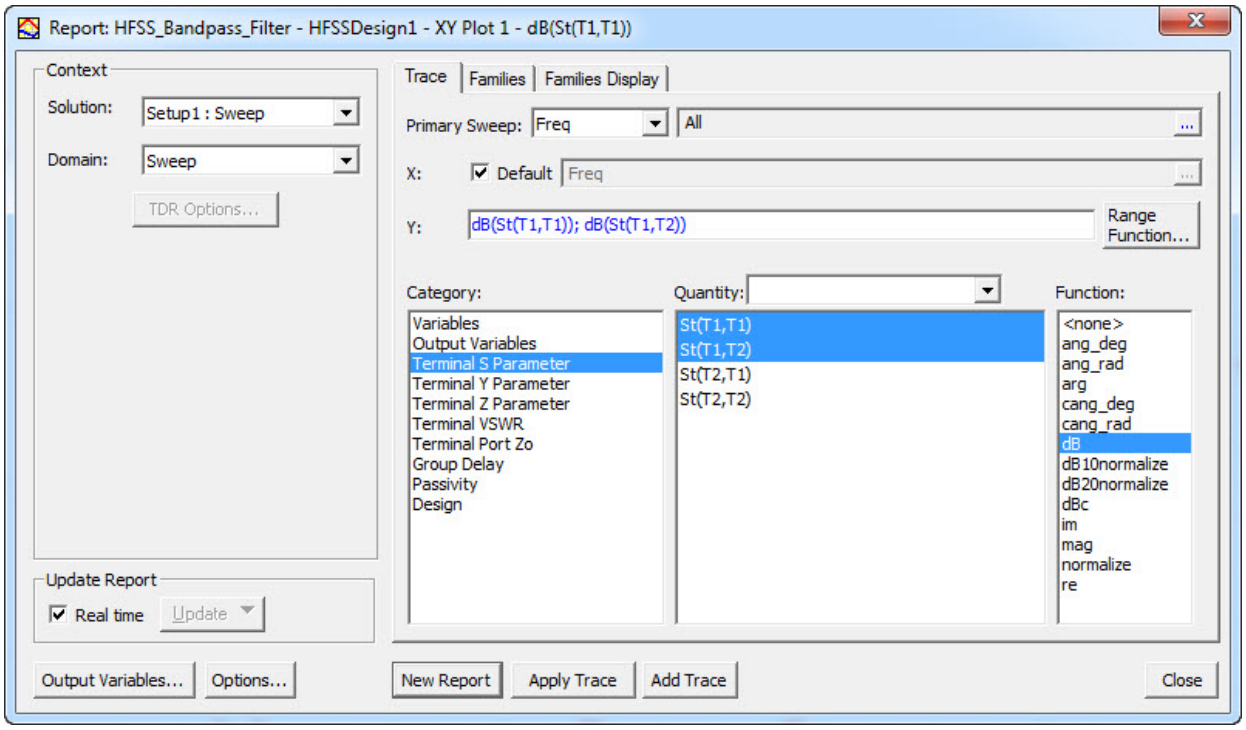


Figure 4-16 Report dialog box

3. Click **New Report.**
4. Double click on the Curve Info text on the legend, and in the Properties dialog, Legend tab, deselect Show Solution Name and Show Variation Key. OK to close the dialog. You can

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

then drag the simplified legend as shown in the figure.

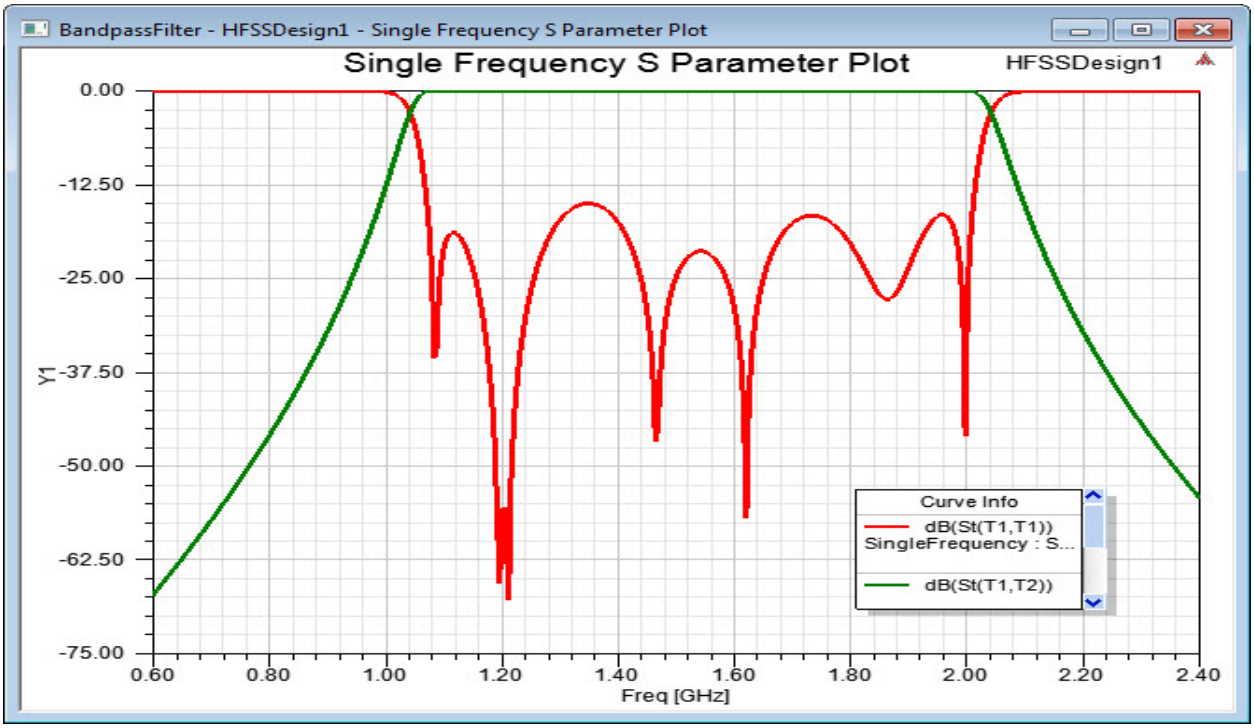


Figure 4-17 S Parameter and Frequency plot

4. Click **Close**.

   The figure shows the characteristics of a typical bandpass filter with almost 0dB insertion loss and reflection of less than -15dB in the pass-band.

Compare S12 with S21

You can verify the symmetry of the bandpass filter by comparing S(T1,T2) with S(T2,T1). To do this, modify the existing S-parameter plot as shown below.

1. Right-click the existing plot and select **Modify Report** from the short-cut menu.

   The **Reports** dialog box appears.

2. Enter the following fields:

- **Solution:** *Setup1: Sweep1*
- **Domain:** *Sweep*
- **Quantity:** *S(T2,T1)*
- **Function:** *dB*

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

3. Click **Add Trace** and click **Close**.

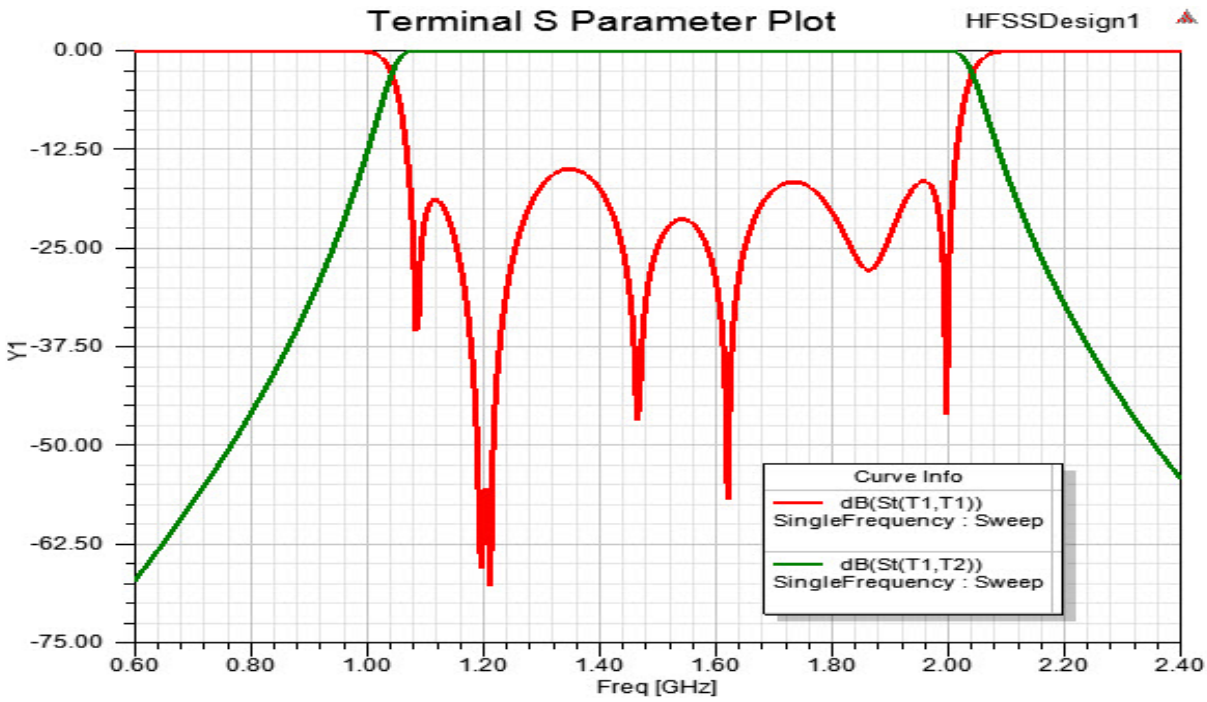


Figure 4-18 S Parameter vs Frequency

Figure 4-19 The curves dB(St(T2,T1) and dB(St(T1,T2) coincide. By hovering the cursor over the option in the Curve info legend, you can highlight the curve of interest.

Change Plot Scale

In order to see the noise in the insertion loss you can zoom in to the plot where the S12 curve plateaus.

To zoom into the area of interest on the plot, change the scale of the Y axis as shown below.

1. Double click the Y-axis.

   The **Y Axis Properties** dialog box appears.
2. On the **Y1 Scaling** tab enter as follows and click **OK**.

- **Min:***-1.0*
- **Max:***0.0*

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

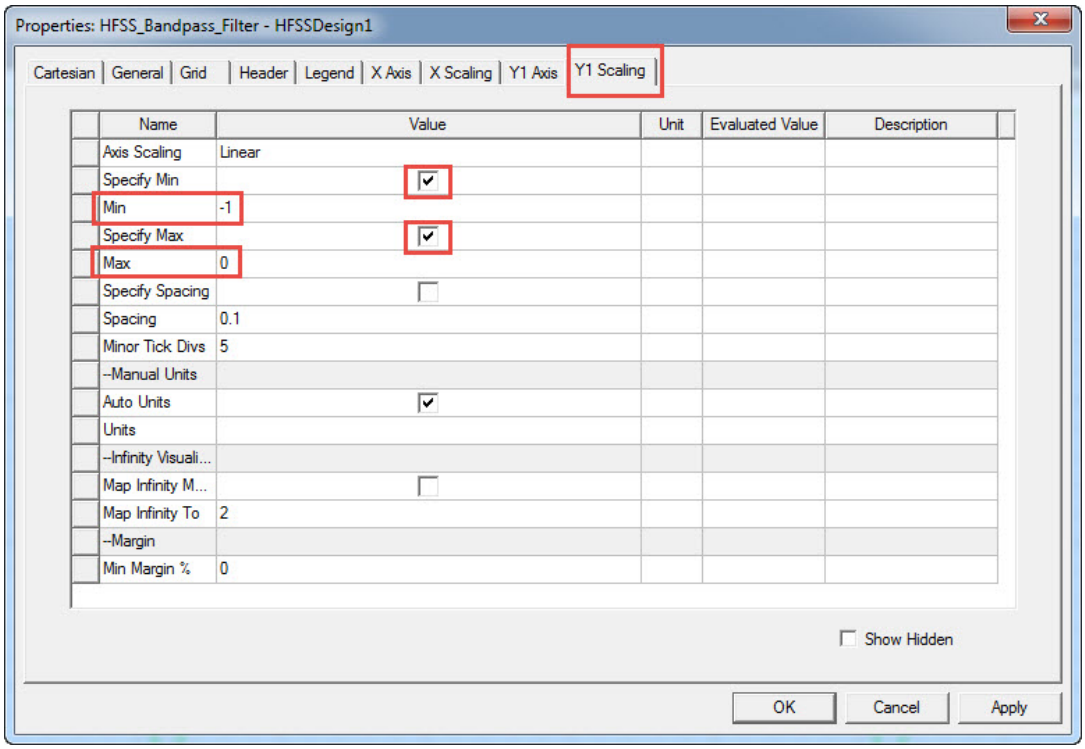
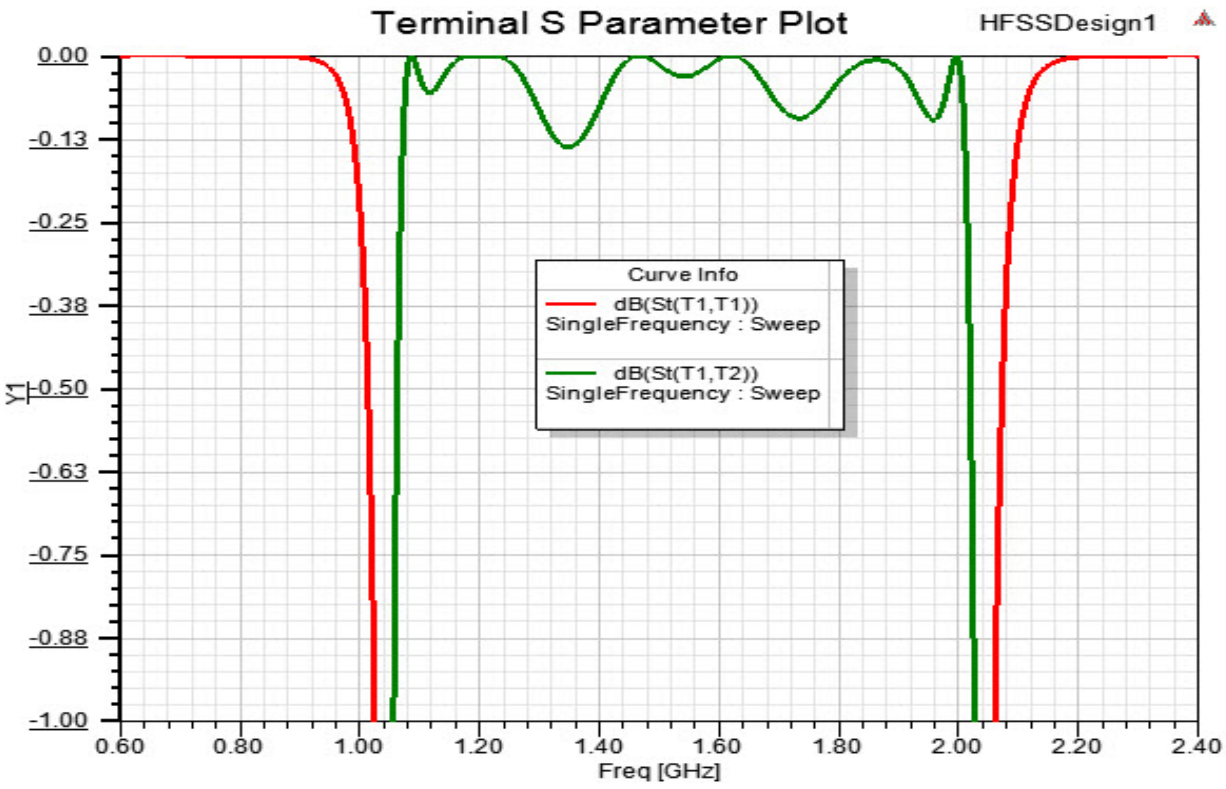


Figure 4-20 Y1 Scaling Properties

Figure 4-21



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

Figure 4-22 S Parameter vs Frequency with Modified Scaling

Create Field Overlays

This section describes how to create field overlays.

1. Click **HFSS>Fields>Edit Sources**.

   The **Edit Sources** dialog box appears.

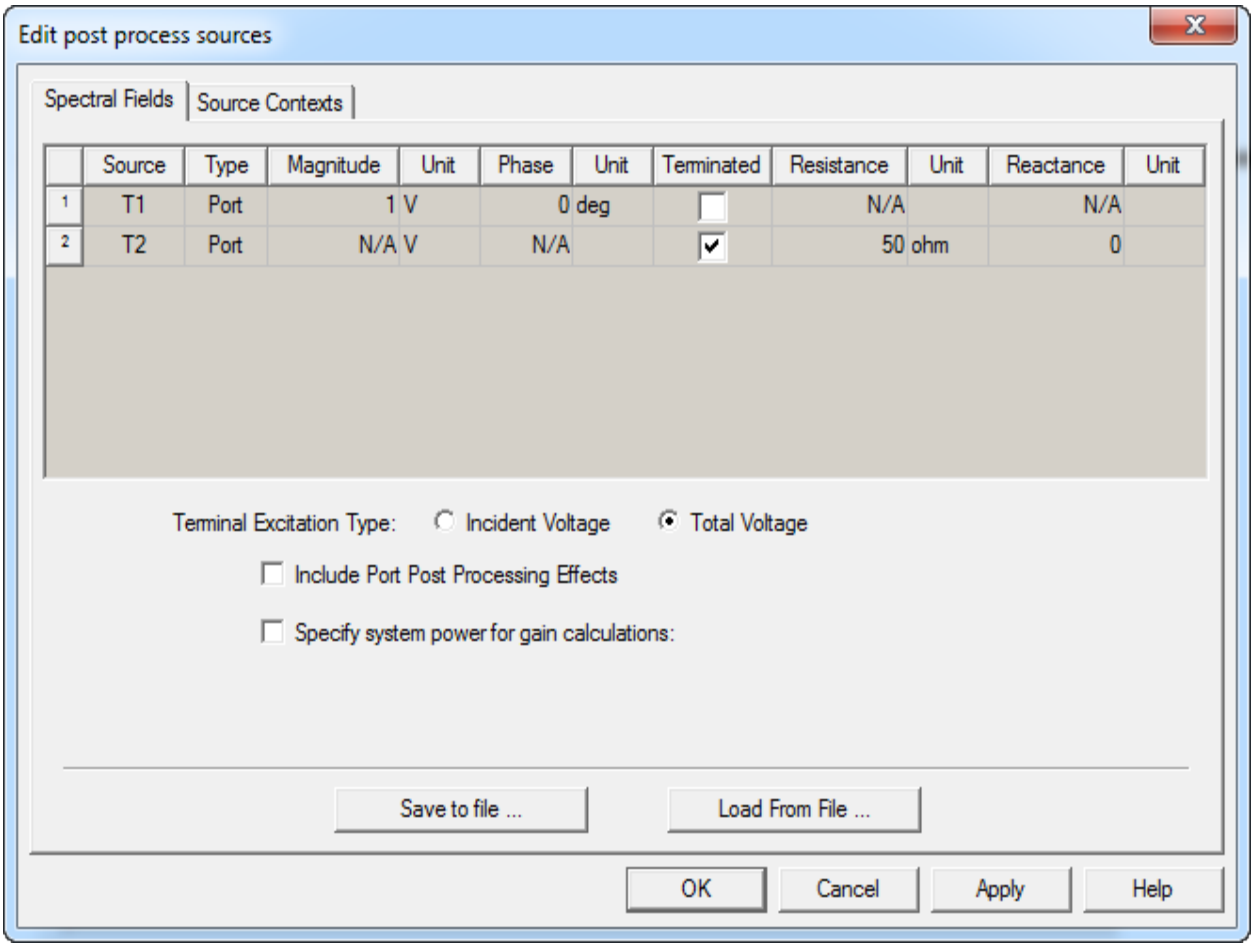


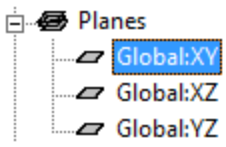
Figure 4-23 Edit Sources dialog box (Total Voltage checked)

2. Set the fields as shown in "Edit Sources dialog box (Total Voltage checked)" above. and click **OK**.

   Note Total Voltage includes both the Incident plus Reflected waves in the calculations.

3. Expand **Planes** from the Model Tree and set **Global:***XY*

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 **HFSS>Fields>Plot Fields>E>Mag_E**
   The **Create Field Plot** dialog box appears.
5. Set the fields as shown in the figure below and click **Done**.

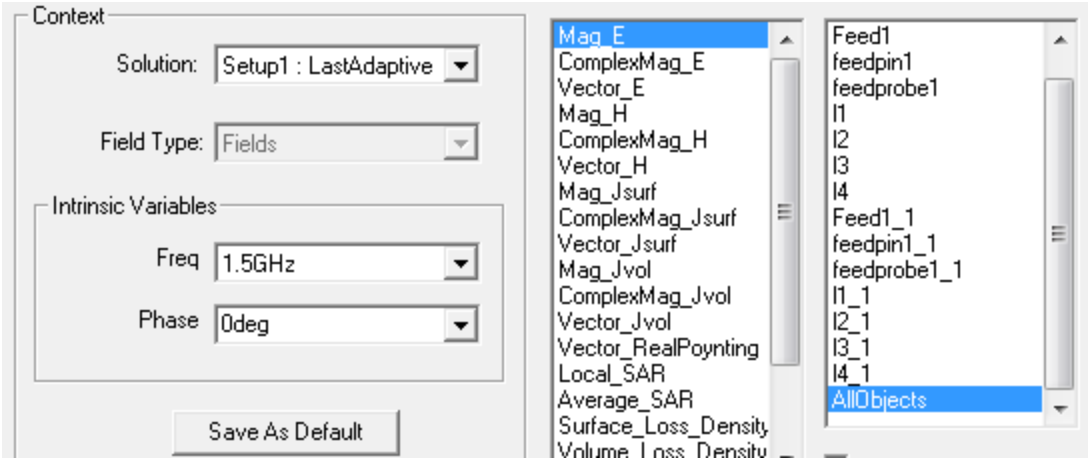


Figure 4-24 Create Field Plot

Modify Plot Attributes

1. Click **HFSS>Fields>Modify Plot Attributes.**
   The **Select Plot Folder** dialog box displays **E Field**.
2. Click **OK**.
   The **E-Field** 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.

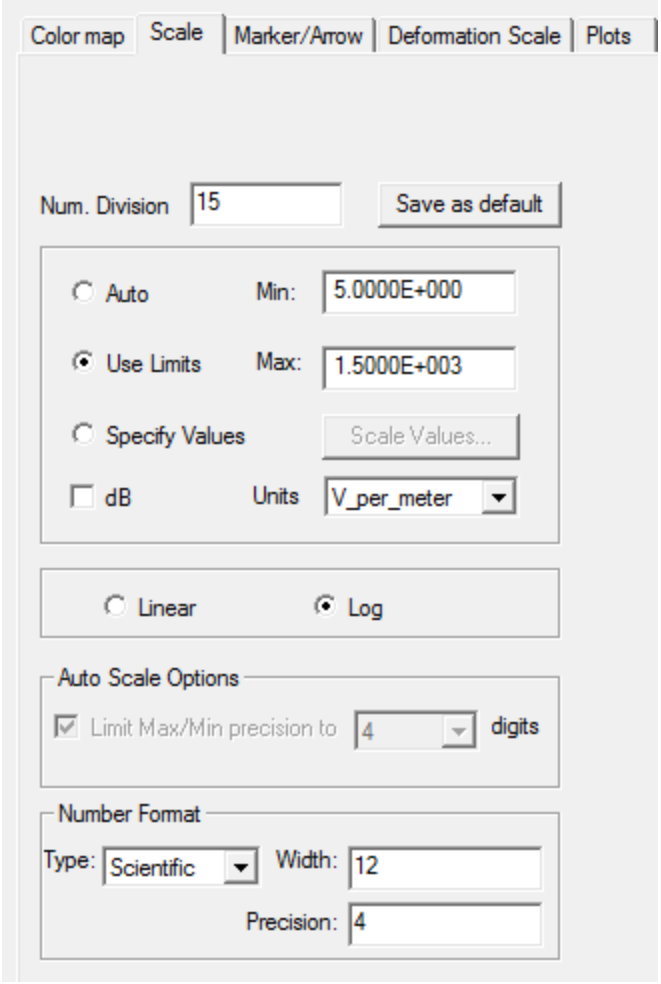


Figure 4-25 E Field dialog box (Scale tab)

3. Click **Scale** and set the fields as shown in "E Field dialog box (Scale tab)" above. and click **Apply**.

   HFSS applies the field overlay 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.

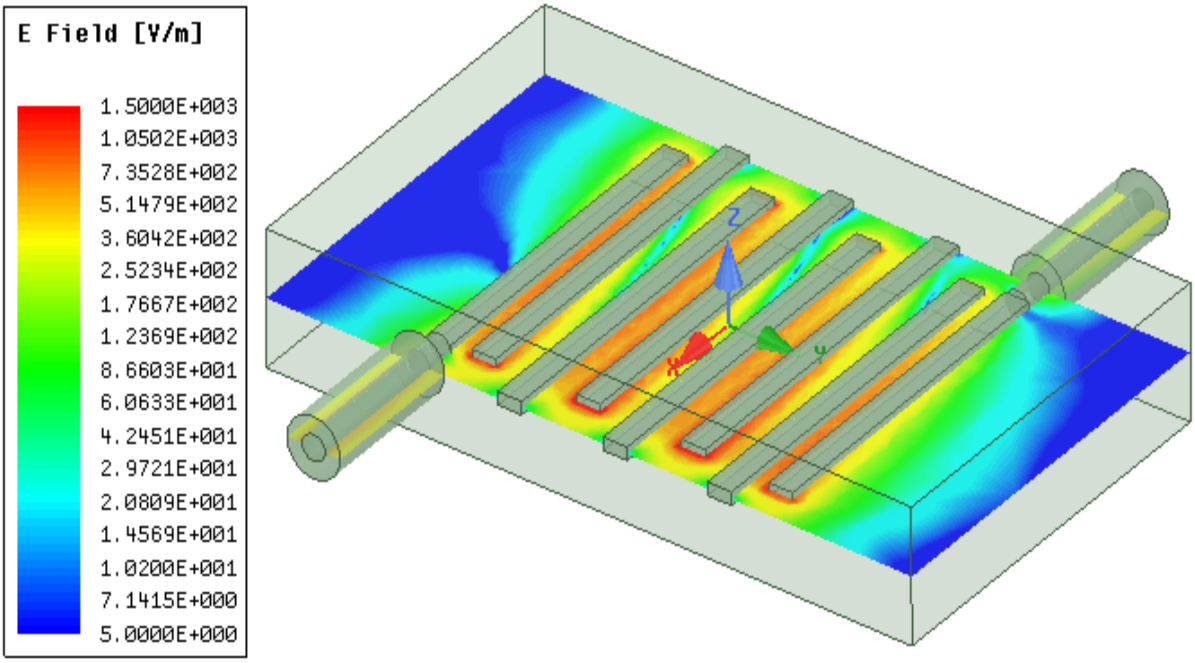


Figure 4-26 With Field Overlay

You may also with to experiment with available settings on the Color Map tab and the Plots tab to see how different settings can modify the appearance of the field plot. You can also select **HFSS>Fields>Animate...** to animate the simulation.

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.