

# Getting Started with HFSS: RCS



February 2009

The information contained in this document is subject to change without notice. Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2009 Ansoft LLC. All rights reserved.

Ansoft LLC  
225 West Station Square Drive  
Suite 200  
Pittsburgh, PA 15219  
USA  
**Phone:** 412-261-3200  
**Fax:** 412-471-9427

HFSS and Optimetrics are registered trademarks or trademarks of Ansoft LLC. All other trademarks are the property of their respective owners.

New editions of this manual incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.


Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

<b>Edition</b>	<b>Date</b>	<b>Software Version</b>
1	February 2009	12

## 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 to type the word **copy**, to type a space, and then to type **file1**.
  - 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*” means to type the word **copy**, to type a space, and then to type a file name.
- 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.

*Alternate methods or tips are listed in the left margin in blue italic text.*

## Getting Help

### Ansoft Technical Support

To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, <http://www.ansoft.com>, click the **Contact** button, and then click **Support**. Phone numbers and e-mail addresses for the technical support staff are listed. You can also contact your Ansoft account manager in order to obtain this information.

All Ansoft 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 HFSS menu bar, click **Help** and select from the menu:

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

### Context-Sensitive Help

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

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

# Table of Contents

## 1. Introduction

RCS Model .....	1-2
General Procedure .....	1-4

## 2. Create the RCS Model

Create the New Project .....	2-2
Add the New Project .....	2-2
Insert an HFSS Design .....	2-2
Add Project Notes .....	2-3
Save the Project .....	2-3
Select the Solution Type .....	2-5
Set Up the Drawing Region .....	2-5
Coordinate System Settings .....	2-5
Units Settings .....	2-6
Create the Geometries .....	2-7
Create the Target Box .....	2-7
Set the Properties for the Target Box .....	2-7
Create the Air Box .....	2-8
Set the Properties for the Air Box .....	2-8
Create the PML Boundaries .....	2-9
Seed the Mesh on the Airbox .....	2-13

Add the Incident Plane Wave .....	2-14
<b>3. Set Up and Generation Solutions</b>	
Add a Solution Setup to the Design .....	1-2
Validate the Design .....	1-3
Analyze the Design .....	1-3
View the Solution Data .....	1-4
View the Profile Data .....	1-4
<b>4. Post Processing for RCS</b>	
Creating the Far Field Infinite Sphere Setups ..	1-2
Creating the Monostatic Setup .....	1-3
Creating the Bistatic Setup .....	1-5
Creating a Far-Field Plot for Bistatic RCS .....	1-6
Creating a Plot for Monostatic RCS .....	1-9

# 1

## Introduction

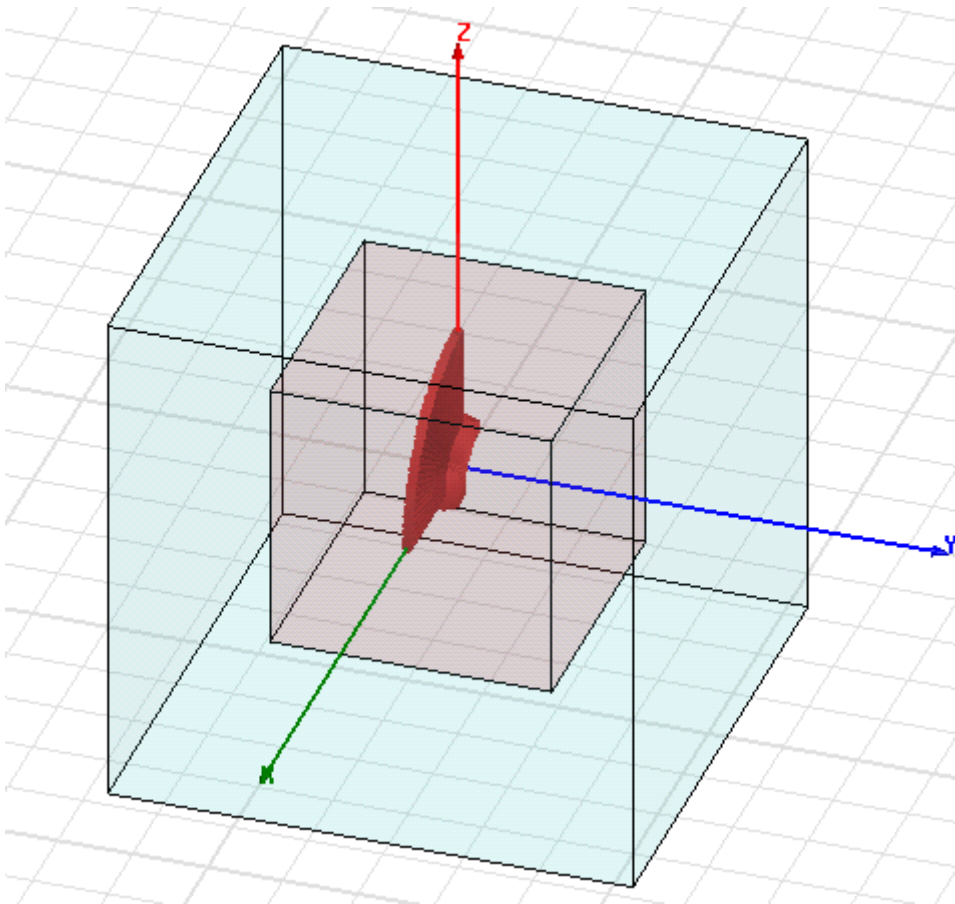
This *Getting Started* guide is written for HFSS users who are modeling Radar Cross Section (RCS) in version 12 for the first time. This guide leads you step-by-step through creating, solving, and analyzing the results of a model that computes RCS.

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

- ✓ Draw the geometric models.
- ✓ Create the Perfectly Matched Layer (PML) Boundaries
- ✓ Add the Excitation
- ✓ Setup Mesh Operations
- ✓ Specify solution setting for the design.
- ✓ Validate the design setups.
- ✓ Run HFSS simulations.
- ✓ Create 2D x-y plots.

## RCS Model

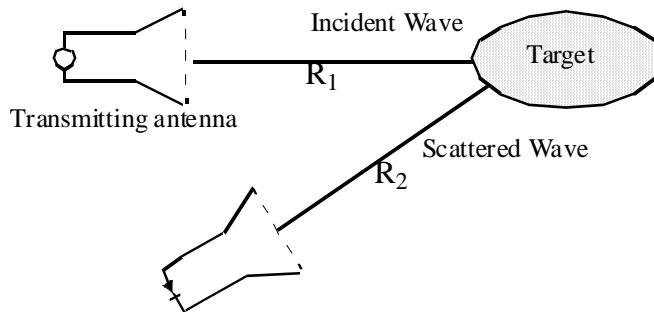
The model for this simulation consists of a perfect electric conducting (pec) target cube surrounded by an airbox. The airbox is surrounded by a PML boundary. The excitation is a regular plane wave. The model has been kept fairly simple, to keep the solution time short. The purpose is to illustrate the basic principles in setting up this kind of problem, and to demonstrate post processing for the RCS information.



The radar cross-section (RCS) or echo area,  $\sigma$ , is measured in meters squared and represented for a bistatic arrangement (that is, when the transmitter and receiver are in different locations as shown in the linked figure).



The following diagram shows the bistatic RCS concept, with separate transmitting and receiving antennas.



HFSS supports RCS for Bistatic, Normalized Bistatic, Complex Bistatic, and Monostatic conditions. In this tutorial, you will generate plots for Normalized Bistatic and Monostatic situations.

## General Procedure

The general procedure for creating and analyzing this RCS project is summarized in the following list:

- 1** Create a project for HFSS.
  - a. Open a new project.
  - b. Add an HFSS design into the new project.
- 2** Draw the geometric model; in this case, a target, and a surrounding airbox that is at least  $\lambda/4$  from the target.
  - a. Set up the drawing region.
  - b. Create the objects that make up the RCS model.
  - c. Assign materials to the objects. in this case pec for the target and vacuum for the air box.
- 3** Set up the problem:
  - a. Set up the PML boundary conditions.
  - b. Set up the plane wave excitation.
- 4** Generate a solution:
  - a. Set up the solution criteria and refine the mesh.
  - b. Generate the solution.
- 5** Use Post Processing to Analyze the RCS solution.

# 2

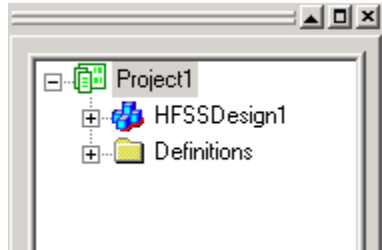
## Create the RCS Model

This section shows how to create the simple RCS model. The major steps are as follows.

- ✓ Create a New Project
- ✓ Create the Geometries
- ✓ Create the PML Boundaries
- ✓ Seed the Mesh
- ✓ Add the Incident Plane wave.

## Create the New Project

The first step in using HFSS to solve a problem is to create a project in which to save all the data associated with the problem. By default, opening HFSS 11 creates a new project named *Project $n$*  and inserts a new project named *HFSS-Design $n$* , where  $n$  is the order in which each was added to the current session.



You can also create a new project and insert a design manually as follows.

## Add the New Project

To add a new HFSS project:

- Click **File>New**.

A new project is listed in the project tree in the **Project Manager** window. It is named *project $n$*  by default, where  $n$  is the order in which the project was added to the current session. Project definitions, such as boundaries and material assignments, are stored under the project name in the project tree.

## Insert an HFSS Design

The next step for this waveguide combiner problem is to insert an HFSS design into the new project. By default, a design named *HFSSDesign $n$*  with the type as [Driven Modal] appears for the current project.

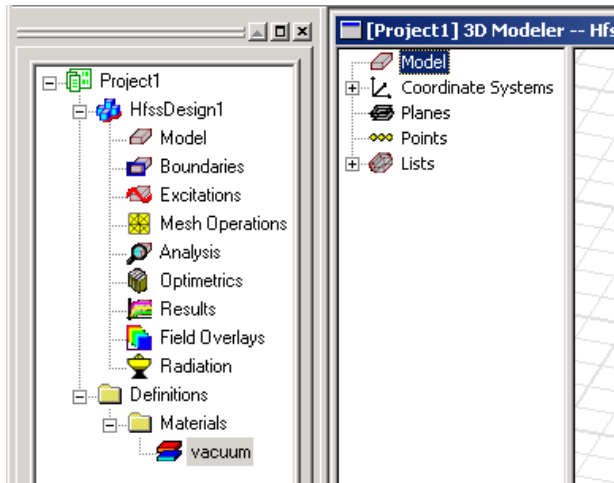
To manually insert an HFSS design into the project, do one of the following:

- Click **Project>Insert HFSS Design**.
- Right-click on the project name in the **Project Manager** window, and then click **Insert>Insert HFSS Design** on the

shortcut menu.

- Click the Insert HFSS Design toolbar button .

A 3D Modeler window appears on the desktop and an HFSS Design icon is added to the project tree, as shown below:



## Add Project Notes

Next, enter notes about your project, such as its creation date and a description of the device being modeled. This is useful for keeping a running log on the project.

To add notes to the project:

- 1 Click HFSS>Edit Notes.

The Design Notes window appears.

- 2 Click in the window and type your notes, such as a description of the model and the version of HFSS in which it is being created.

- 3 Click OK to save the notes with the current project.

**Note** To edit existing project notes, double-click **Notes** in the project tree. The Design Notes window appears, in which you can edit the project's notes.

## Save the Project

Next, save and name the new project.

It is important to save your project frequently. Depending on the setting in **Tools>Options>HFSS General Options** dialog, **Project Options** tab, HFSS can automatically save models at specified intervals.

To save the new project:

**1** Click **File>Save As**.

The **Save As** dialog box appears.

**2** Use the file browser to find the directory where you want to save the file.

**3** Type the name `rcs_example` in the **File name** text box.

**4** In the **Save as type** list, click **Ansoft HFSS Project (.hfss)** as the correct file extension for the file type.

When you create an HFSS project, it is given a `.hfss` file extension by default and placed in the **Project** directory. Any files related to that project are stored in that directory.

**5** Click **Save**.

HFSS saves the project to the location you specified.

**Note** For further information on any topic in HFSS, such as coordinate systems and grids or 3D Modeler commands or windows, you can view the context-sensitive help:

- Click the **Help** button in a pop-up window.
- Press **Shift+F1**. The cursor changes to `?`. Click on the item with which you need help.
- Press **F1**. This opens the Help window. If you have a dialog open, the Help opens to a page that describes the dialog.
- Use the commands from the **Help** menu.

## Select the Solution Type

Before you draw the RCS model, first you must specify a solution type. As you set up your model, available options will depend on the design's solution type.

To specify the solution type:

- 1 Click **HFSS>Solution Type**.

The **Solution Type** window appears.

- 2 This antenna project is a mode-based problem; therefore, select the **Driven Modal** solution type.

The possible solution types are described below.

**Driven Modal** For calculating the mode-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines, which are “driven” by a source, and for computing incident plane wave scattering.

**Driven Terminal** For calculating the terminal-based S-parameters of passive, high-frequency structures with multi-conductor transmission line ports, which are “driven” by a source. Results in a terminal-based description in terms of voltages and currents.

**Eigenmode** For calculating the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

- 3 Click **OK** to apply the **Driven Modal** solution type to your design.

## Set Up the Drawing Region

The next step is to set up the drawing region. For this RCS problem, you decide the coordinate system, and specify the units and grid settings.

## Coordinate System Settings

For this RCS problem, you will use the fixed, default global coordinate system (CS) as the working CS. This is the current CS with which objects being drawn are associated.

HFSS has three types of coordinate systems that let you easily orient new objects: a *global* coordinate system, a *relative* coordinate system, and a *face* coordinate system. Every CS has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes.

## Units Settings

Now, specify the drawing units for your model. For this antenna problem, set the drawing units to meter.

To set the units:

- 1** Click **Modeler>Units**.

The **Set Model Units** dialog box appears.

- 2** Select **meter** from the **Select units** menu. Make sure **Rescale to new units** is cleared.

If selected, the **Rescale to new units** option automatically rescales the grid spacing to units entered that are different than the set drawing units.

- 3** Click **OK** to accept meters as the units for this model.



## Create the Geometries

The geometries for this RCS model consists of the basic objects listed below with their dimensions:

<b>target</b>	A pec box .75 meter square. At the 300 Mhz Frequency we use in the simulation, this is $0.75\lambda$
<b>air box</b>	A vacuum box 1.4 meter square. This meets the requirement that PML boundaries should be at least $\lambda/4$ from the target.

### Create the Target Box

To create the target box, use the **Draw>Box** command to create a random box, and edit the properties for the following position and dimensions:

Coordinates	-0.375, -0.375, -0.375
XSize	0.75
YSize	0.75

### Set the Properties for the Target Box

To set the properties for the box:

- 1** Select the newly created box in the history tree, and right click **Properties** from the shortcut menu.  
This displays the **Properties** dialog.
- 2** Edit the name field to target.
- 3** In the materials field, press the button to display the **Materials library** dialog.
- 4** Select pec from the materials list, and click **OK** to close the dialogue.
- 5** In **Properties** dialogue for the box, **Edit** the color as a dark red.
- 6** Set the transparency as 0.6.
- 7** Click **OK** to accept the settings and close the dialog.

## Create the Air Box

To create the air box, use the **Draw>Box** command to create a random box, and edit the properties for the following position and dimensions:

Coordinates	-0.7, -0.7, -0.7
XSize	1.4
YSize	1.4

These dimensions will give the air box a suitable distance from the target, greater than  $\lambda/4$  wavelength on each side, relative to the 300 Mhz frequency we will use.

## Set the Properties for the Air Box

To set the properties for the air box:

- 1** Select the newly created box in the history tree, and right click **Properties** from the shortcut menu.

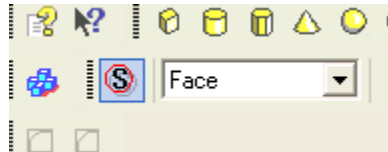
This displays the **Properties** dialog.

- 2** Edit the name field to `air_box`.
- 3** In the materials field, press the button to display the **Materials library** dialog.
- 4** Select vacuum from the materials list, and click **OK** to close the dialogue.
- 5** In **Properties** dialogue for the box, Edit the color as a light blue.
- 6** Set the Transparency as 0.8.
- 7** Click **OK** to accept the settings and close the dialog.

## Create the PML Boundaries

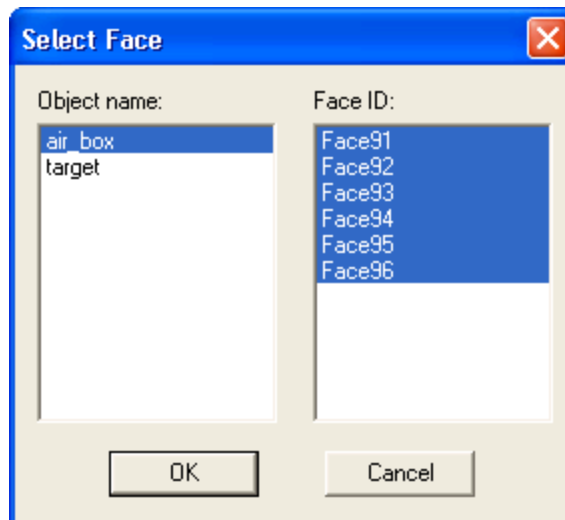
To create the PML boundaries:

- 1 Set the selection options to Face, either with the menu command **Edit>Select>Face**, the toolbar drop down menu for Face, or the F quick key.
- 2 Select **Edit>Select>By Name**, or click the select icon in the toolbar.



This displays the Select by Face dialog.

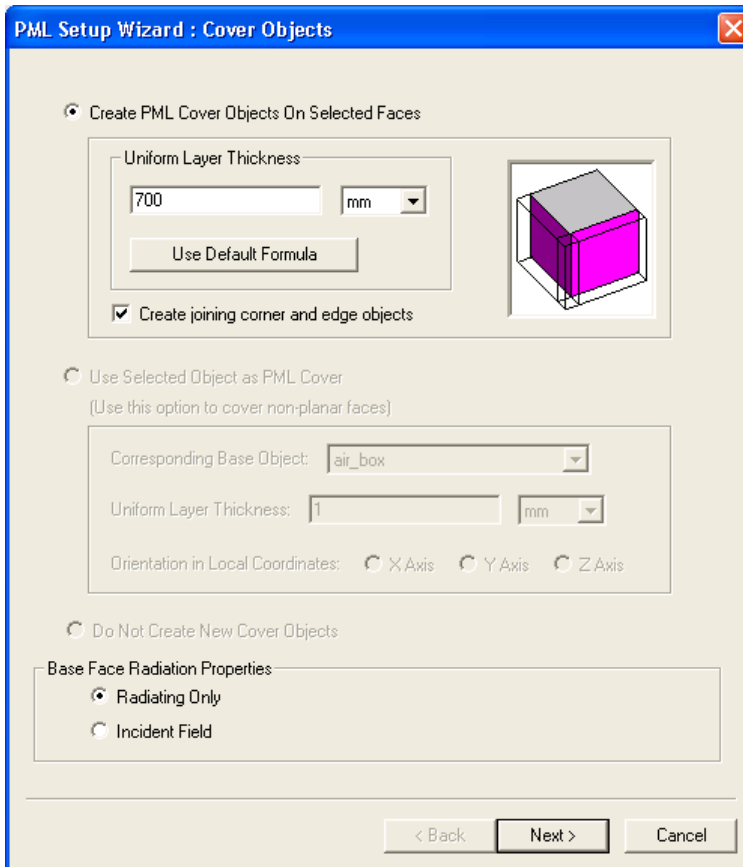
- 3 From the Object list, select `air_box`.  
This lists the names of the `air_box` faces.
- 4 Hold down the Ctrl key, and click each face.



All faces of the airbox should be highlighted.

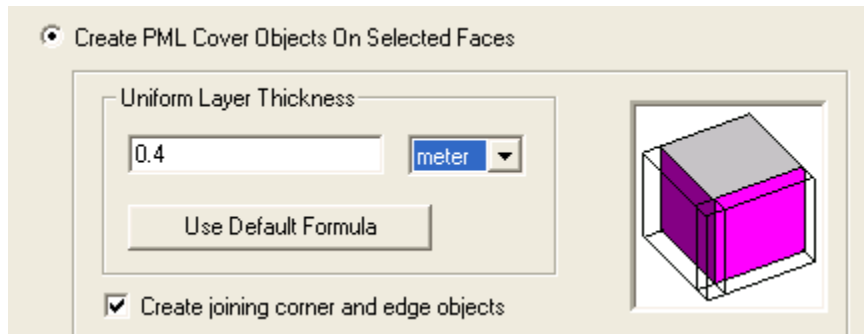
- 5 In the Modeler window, right click to display the shortcut menu, and select **Assign Boundary> PML Setup Wizard**.

The setup wizard displays.



- 6 In the Uniform Layer Thickness field, set the thickness to 0.4 meter. This will keep the solution small enough for this exercise. The layers' material parameters will be adjusted

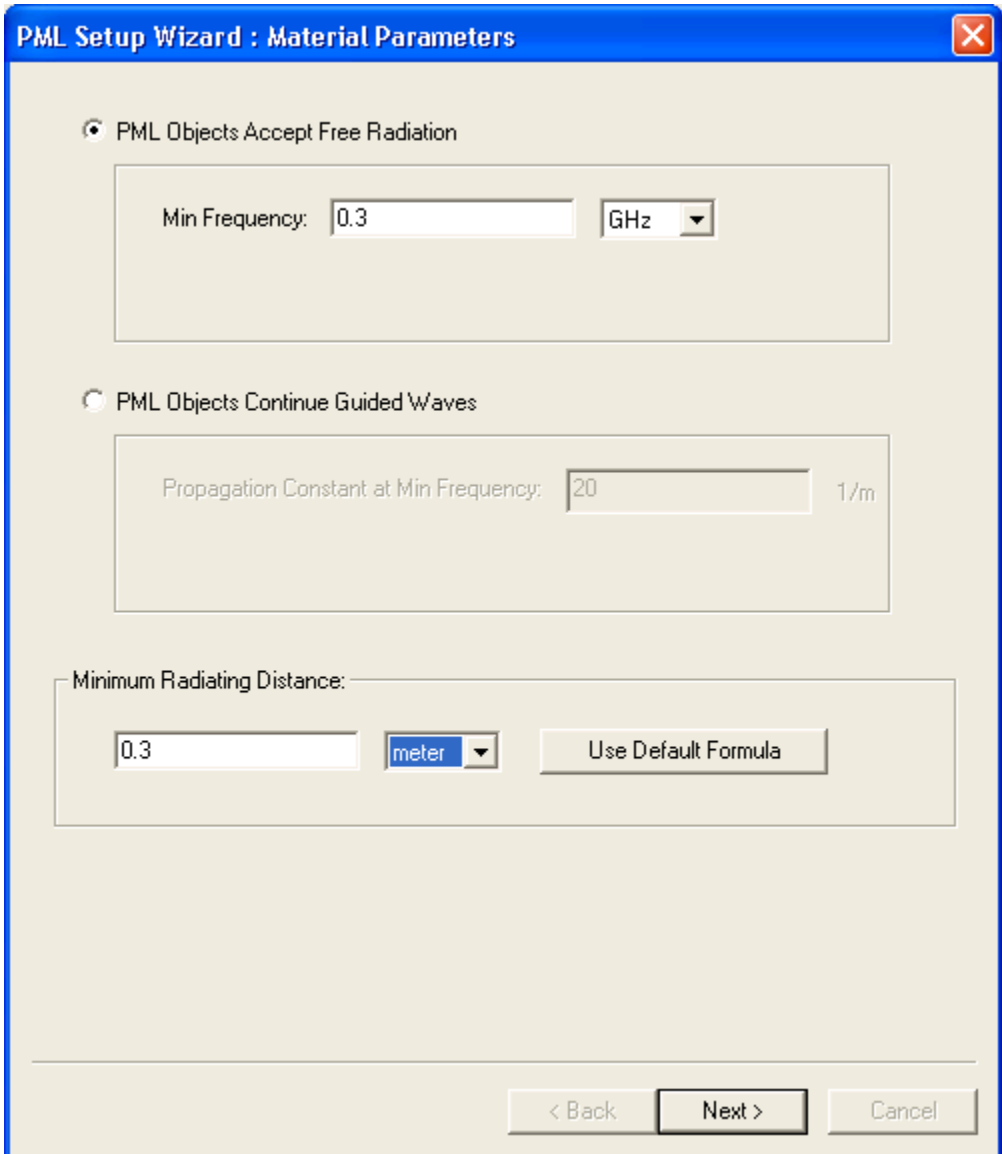
automatically in accordance with the new thickness.



- 7 Leave the Create joining corner and edge objects check-box selected, and click **Next**.

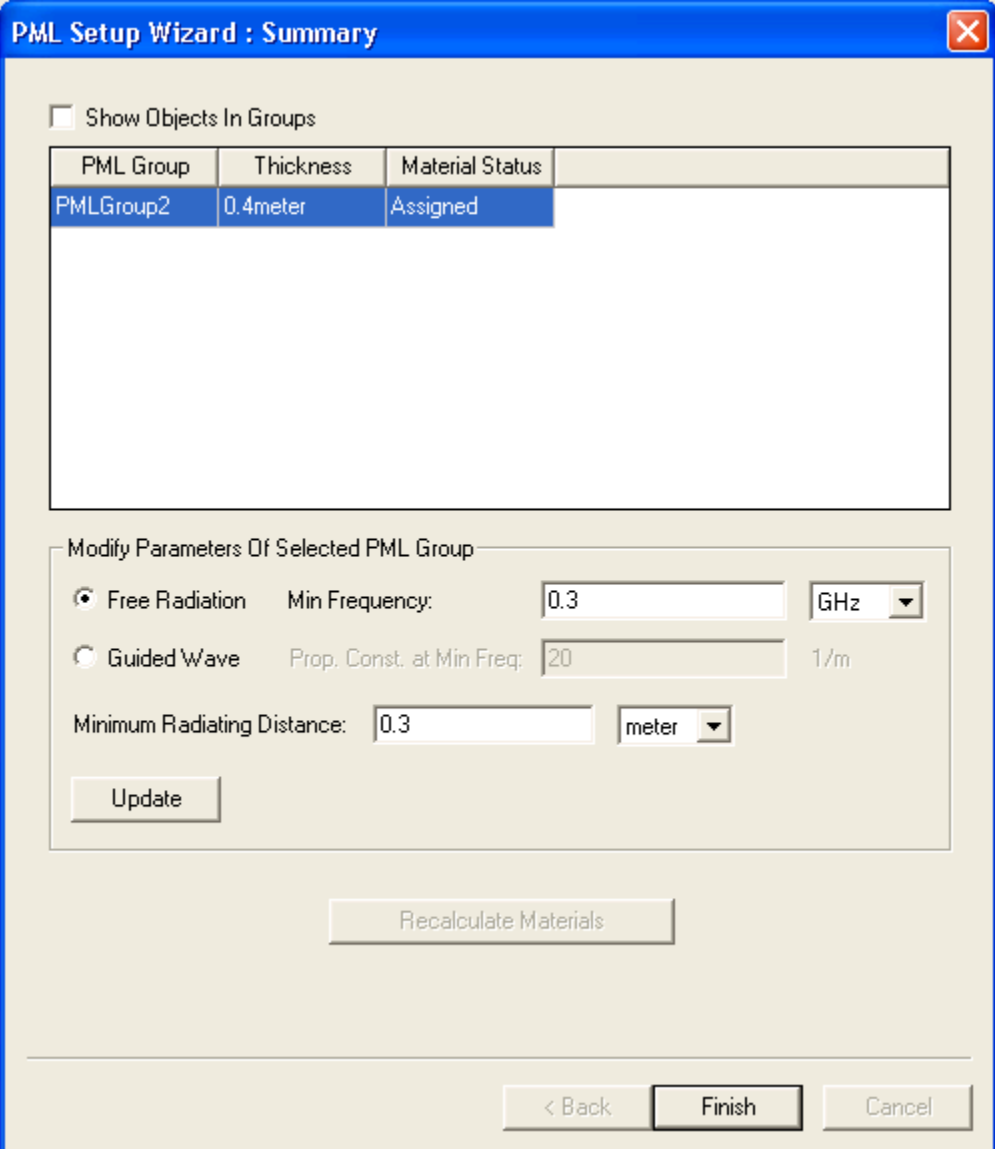
This creates the PML objects, and displays the **Material**

Parameters dialog.



- 8 Set the Minimum Frequency to 0.3 Ghz, and the Minimum Radiating Distance to 0.3 meter, as shown.

9 Click Next to display the PML Summary dialog.



The image shows the 'PML Setup Wizard : Summary' dialog box. It has a blue title bar with a close button (X) in the top right corner. Below the title bar, there is a checkbox labeled 'Show Objects In Groups' which is currently unchecked. Below this is a table with three columns: 'PML Group', 'Thickness', and 'Material Status'. The table contains one row with the following data: 'PMLGroup2', '0.4meter', and 'Assigned'. Below the table is a section titled 'Modify Parameters Of Selected PML Group'. This section contains three radio buttons: 'Free Radiation' (selected), 'Guided Wave', and 'Prop. Const. at Min Freq.'. The 'Free Radiation' option has a 'Min Frequency' input field with the value '0.3' and a unit dropdown menu set to 'GHz'. The 'Guided Wave' option has a 'Prop. Const. at Min Freq.' input field with the value '20' and a unit dropdown menu set to '1/m'. The 'Prop. Const. at Min Freq.' option has a 'Minimum Radiating Distance' input field with the value '0.3' and a unit dropdown menu set to 'meter'. Below these options is an 'Update' button. At the bottom of the dialog box, there is a 'Recalculate Materials' button and a row of three buttons: '< Back', 'Finish', and 'Cancel'.

PML Group	Thickness	Material Status
PMLGroup2	0.4meter	Assigned

Modify Parameters Of Selected PML Group

Free Radiation    Min Frequency:  GHz

Guided Wave    Prop. Const. at Min Freq:  1/m

Minimum Radiating Distance:  meter

Update

Recalculate Materials

< Back    Finish    Cancel

10 Click Finish to close the dialog.

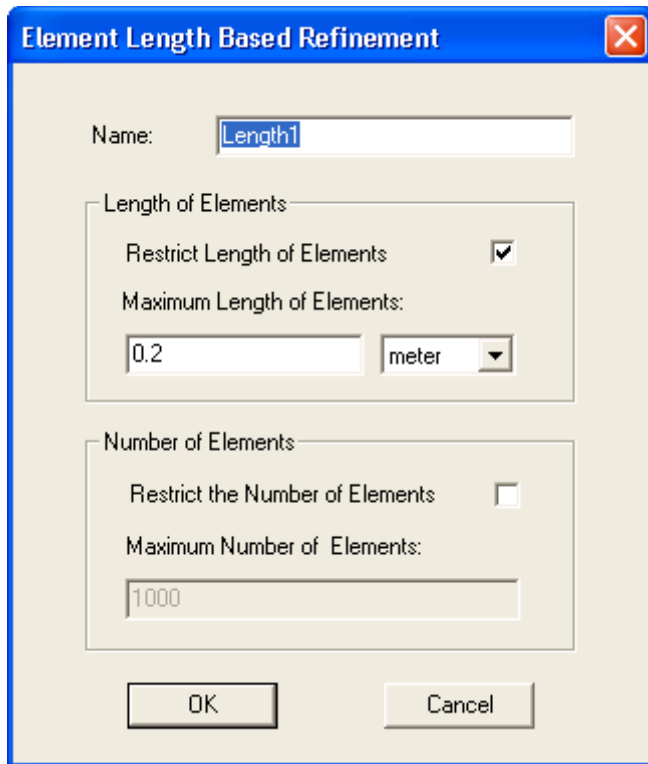
The PML boundaries are listed under Boundaries in the Project tree, and the PML objects are listed in the History tree.

## Seed the Mesh on the Airbox

Seed the mesh on the air\_box to  $\lambda/5$ . This will result in a very accurate radiation pattern.

- 1 Select the faces of the air\_box
- 2 Right click on **Mesh Operations** in the Project tree.
- 3 Click **Assign>On Selection>Length Based**.

This displays the **Element Length Based Refinement** dialog.



- 4 Set the Maximum length of Elements value to 0.2 with the units as meter.
- 5 Click OK to close the dialog.

The Length1 icon appears under **Mesh Operations** in the Project tree.



## Add the Incident Plane Wave

An incident plane wave is a wave that propagates in one direction and is uniform in the directions perpendicular to its direction of propagation.

- 1** Click **HFSS>Excitations>Assign>Incident Wave>Plane Wave**.

The **Incident Wave Source: General Data** page appears.

- 2** Type the source's name in the **Name** text box or accept the default name.
- 3** Select the **Vector Input Format** as **Spherical** coordinates.
- 4** Enter 0, 0, 0 for the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
- 5** Click **Next**.
- 6** The **Incident Wave Source: Spherical Vector Setup** page appears.
  - a.** Under **IWaveTheta**, enter 0 deg for **Start**, 90 deg for **Stop**, and 3 for **Step**. For the monostatic case, the RCS will be computed only at values of **IWave  $\theta$**  entered here. For the purposes of this demo, this keeps the number points down and to save on the solution time
  - b.** Click **View Point List** to see the values of  $\theta$ .
- 7** Click **Next**. the **Incident Wave Source: Plane Wave Options** page appears.
- 8** Select the **Type of Plane Wave**.
- 9** Select **Regular/Propagating**, so no other fields are active.
- 10** Click **Finish**. The incident wave you defined is added to the **Excitations** list in the Project.



# 3

## Set Up and Generation Solutions


In this chapter you will complete the following tasks:

- ✓ Add a solution setup.
- ✓ Add a frequency sweep to the solution setup.
- ✓ Validate the design.
- ✓ Run the analysis.

## Add a Solution Setup to the Design

Specify how HFSS will compute the solution by adding a *solution setup* to the design.

In the solution setup, you will instruct HFSS to perform an adaptive analysis at 10 GHz. During an adaptive analysis, HFSS refines the mesh iteratively in the areas of highest error.

- 1** In the project tree, under the rcs\_example design, right-click **Analysis**, and then click **Add Solution Setup**  on the shortcut menu.

The **Solution Setup** dialog box appears.

- 2** Under the **General** tab, type **0.3** in the **Solution Frequency** text box, and leave the default unit set to **GHz**.
- 3** Leave the **Maximum Number of Passes** set to **6**. This is the maximum number of mesh refinement cycles that HFSS will perform.
- 4** Leave **Maximum Delta energy** at **0.1**.
- 5** Leave the default settings and click **OK**.

The solution setup is listed in the project tree under **Analysis**. It is named *Setup1* by default.

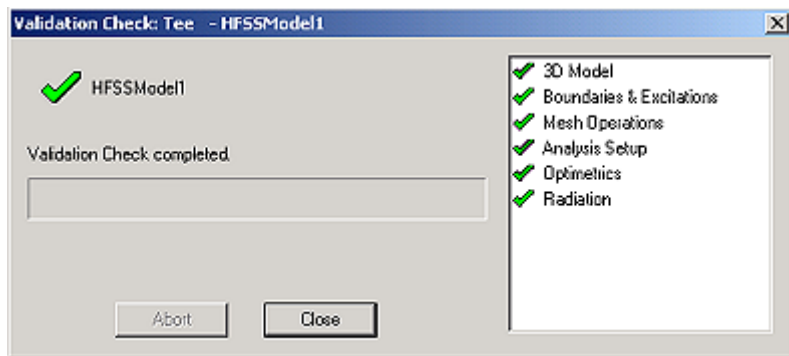


## Validate the Design

Before you run an analysis, it is helpful to verify that all of the necessary setup steps have been completed and their parameters are reasonable.

- 1 On the HFSS menu, click **Validation Check** .

HFSS checks the project setup, and then the **Validation Check** window appears.




- 2 Click **Close**.

Now you are ready to run the simulation.

## Analyze the Design

Now you will run the simulation.

- On the HFSS menu, click **Analyze All** .

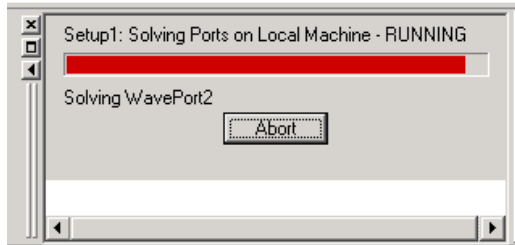
HFSS computes the 3D field solution for every solution setup in the project. In this problem, Setup1 is the only setup.

The solution process is expected to take approximately 5 to 30 minutes, depending on the machine speed and load.

When the solution is complete, a confirmation message appears in the **Message Manager**.

The **Progress** window displays the solution progress as it

OCCURS:



**Note** The results that you obtain should be approximately the same as the ones given in this section. However, there may be a slight variation between platforms.

## View the Solution Data

While the analysis is running, you can view a variety of profile, convergence, and matrix data about the solution.

## View the Profile Data

While the solution proceeds, examine the computing resources, or profile data, used by HFSS during the analysis. The profile data is essentially a log of the tasks performed by HFSS during the solution. The log indicates the length of time each task took and how much RAM/disk memory was required.

# 4

## Post Processing for RCS

This chapter describes how to create the geometry setups for monostatic and bistatic infinite spheres. You can then create plots for these geometries for a Normalized Bistatic RCS and Monostatic RCS. Normalized RCS means RCS normalized with respect to wavelength squared.

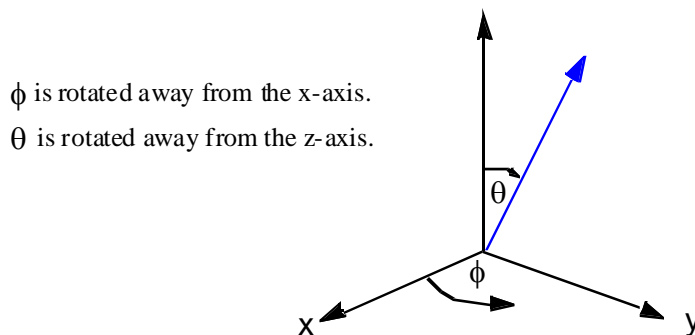
## Creating the Far Field Infinite Sphere Setups

To evaluate radiated fields in the far-field region, you must set up an infinite sphere that surrounds the radiating object. For this example, we will create setups for the bistatic and monostatic cases.

When you set up a spherical surface over which to analyze near or far fields, you specify a range and step size for phi and theta. These indicate the spherical direction in which you want to evaluate the radiated fields. For every value of phi there is a corresponding range of values for theta, and vice versa. This creates a spherical grid. Each grid point indicates a unique direction along a line that extends from the center of the sphere through the grid point. The radiated field is evaluated in this direction. The number of grid points is determined by the step size for phi and theta.

The sphere can be defined according to any defined coordinate system and before or after a solution has been generated.

The relationship between phi and theta is shown below.



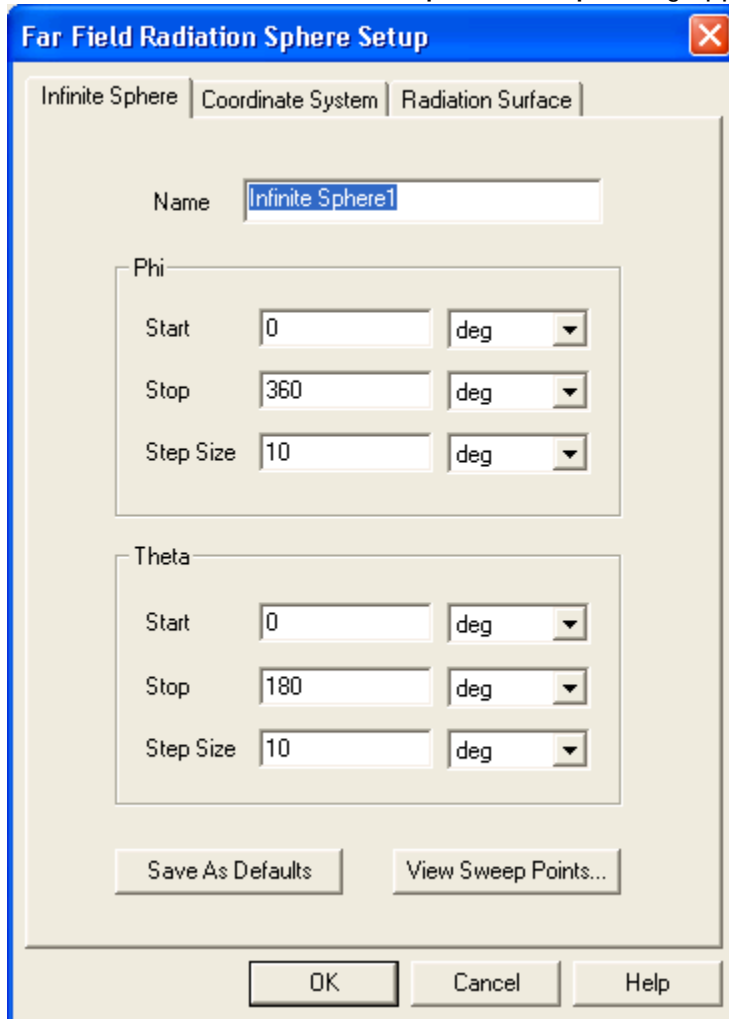
When HFSS evaluates the radiated fields, it needs at least two directions along which to plot the fields. Therefore, if the step size for phi is zero, then the step size for theta must be greater than zero, and vice versa. This ensures that the fields are plotted in at least two directions.



## Creating the Monostatic Setup

- 1 Click HFSS>Radiation>Insert Far Field Setup>Infinite Sphere.

The Far Field Radiation Sphere Setup dialog appears.



- 2 Under the **Infinite Sphere** tab, type a name for the sphere in the **Name** text box.  
For the monostatic sphere, type the name **monostatic**.
- 3 Specify the range of angles to include in the sphere. For the monostatic case, phi and theta will be dummy values.

The reason is that when we ask, later, for a plot of monostatic RCS, the software will know in which direction to compute the far field for every incident angle:  $i\omega\theta$  and  $i\omega\phi$  from the excitation setup.

- a. Specify the following for **Phi**, in degrees (**deg**) or radians (**rad**):

<b>Start</b>	The point where the rotation of phi begins. Leave this as 0.
<b>Stop</b>	The point where the rotation of phi ends. The <b>Stop</b> value must be greater than or equal to the <b>Start</b> value and less than 360. If the <b>Stop</b> value is equal to the <b>Start</b> value, then HFSS assumes that only one angle should be used and the <b>Step Size</b> value will be ignored. Set this to 0.
<b>Step Size</b>	The number of degrees or radians (spherical grid points) between the sweep of phi. Entering zero for the <b>Step Size</b> causes the sweep to consist of one point, the start value. If the <b>Step Size</b> value is zero, then HFSS assumes that only one angle should be used. Set this to 0.

- b. Specify the following for **Theta**, in degrees (**deg**) or radians

(rad):

- |                  |  |
|------------------|--|
| <b>Start</b>     | The point where the rotation of theta begins. The <b>Start</b> value must be greater than -90 degrees, or the equivalent in radians.<br>Set this to 0.   |
| <b>Stop</b>      | The point where the rotation of theta ends. The <b>Stop</b> value must be greater than the <b>Start</b> value and less than 90 degrees, or the equivalent in radians. If the <b>Stop</b> value is equal to the <b>Start</b> value, HFSS assumes that only one angle should be used and the <b>Step Size</b> value will be ignored.<br>Set this to 0. |
| <b>Step Size</b> | The number of degrees or radians (spherical grid points) between the sweep of theta.<br>Entering zero for the number of steps causes the sweep to consist of one point, the <b>Start</b> value. If the <b>Step Size</b> value is zero, then HFSS assumes that only one angle should be used.<br>Set this to 0.                                       |

- 4 Click the **Coordinate System** tab, and then to orient the sphere according to the global coordinate system (CS), select **Use global coordinate system**.
  - If you needed to orient the sphere according to a user-defined CS, you would select **Use local coordinate system** and then select a defined CS from the **Choose from existing coordinate systems** list.
- 5 Click the **Radiation Surface** tab.  
Leave the selection as **Use Boundary Radiation Surfaces**. If you needed to specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, you could select **Use Custom Radiation Surface**.
- 6 Click **OK**.  
The monostatic infinite sphere is created. It is listed in the

project tree under **Radiation**.

**Note** You must have defined at least one radiation or PML boundary in the design for HFSS to compute far-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the far fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Far Field Radiation Sphere Setup** window.

## Creating the Bistatic Setup

- 1** In the Project tree, right-click on Radiation and from the short cut menu click **Insert Far Field Setup>Infinite Sphere**.

The **Far Field Radiation Sphere Setup** dialog appears.

- 2** This time specify the name as bistatic.
- 3** Set the Phi Start, Stop, and Step values as 0.
- 4** Set the Theta Start value as 0, the Stop value as 180 deg and the Step values as 1.
- 5** Click **OK** to close the dialog.

The bistatic setup appears under **Radiation** in the Project tree.

## Creating a Far-Field Plot for Bistatic RCS

First, create a plot for bistatic RCS.

To select a far-field quantity to plot:

- 1** To create the report, right-click on the Results icon in the Project tree, and select **Create Far Fields Report>Rectangular Plot**.

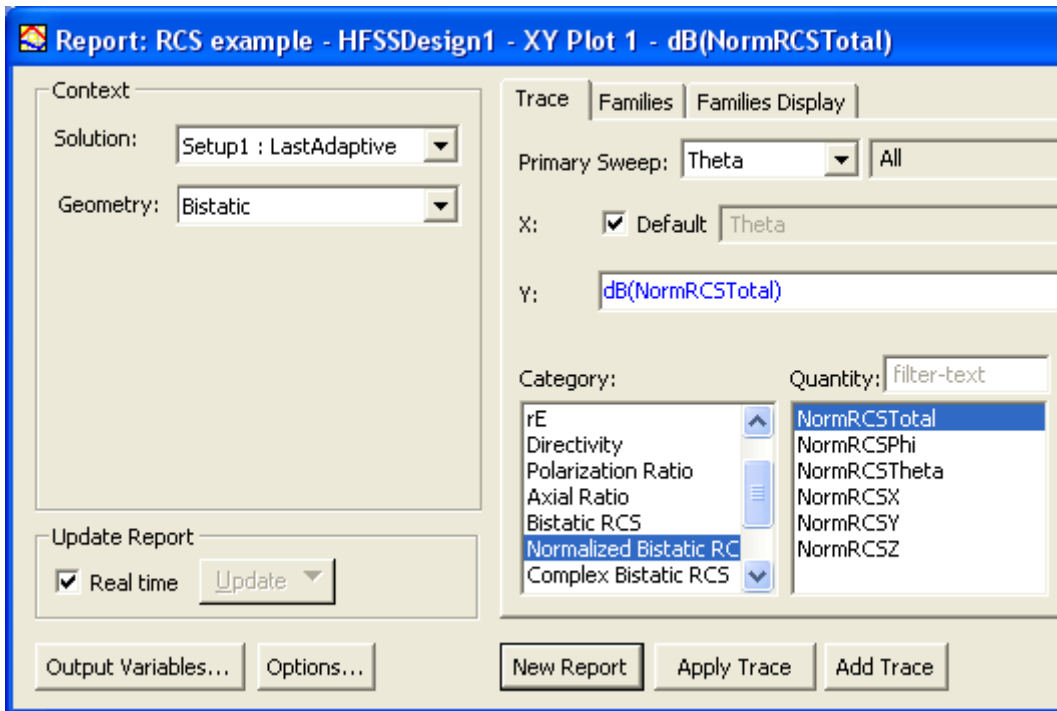
This displays the **New Report - New Traces** dialog.

- 2** Leave the Context selection for the Solution as Setup1: LastAdaptive.
- 3** In order to plot the RCS, you select one of the geometries you created from the **Geometry** list in the Traces dialog. For this plot, select **Bistatic** as the Geometry.
- 4** For Category, select **Normalized Bistatic RCS**.  
This selection causes the **Quantity** list to show NormRCS quantities for Total, Phi, Theta, X, Y, and Z, with Total Selected.

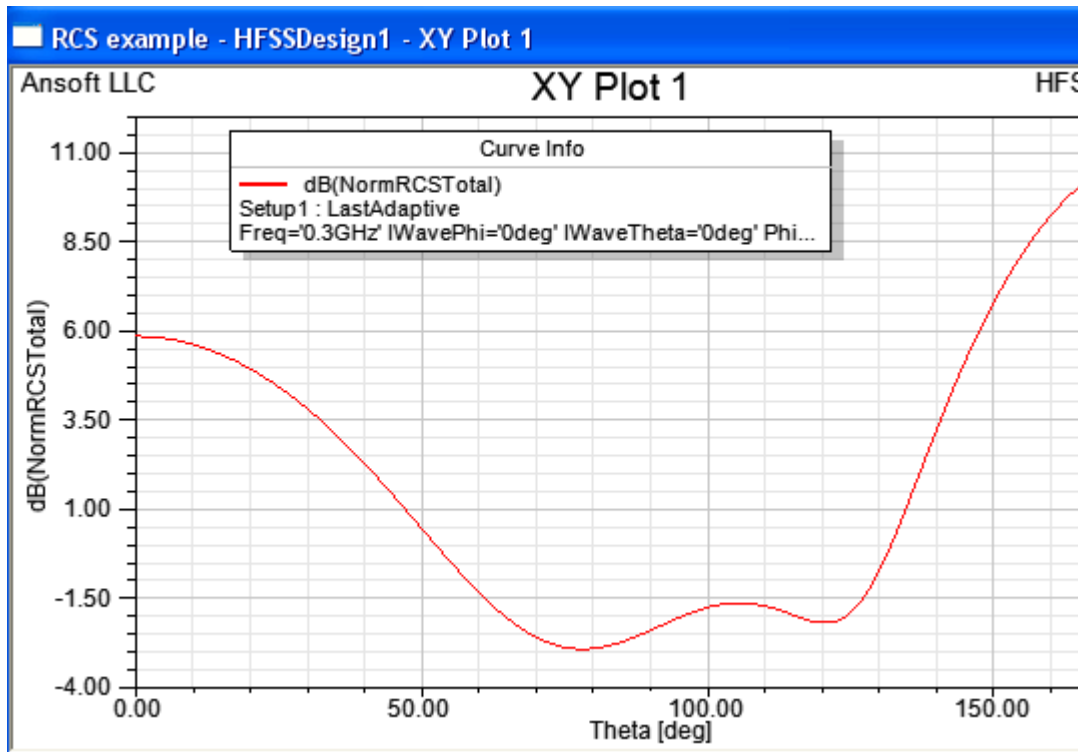
- 5** For the function, select **dB** function.

After these sections, the Y field under Trace shows db(NormRCSTotal). The X value will be the Theta for the

incident wave defined for the simulation



- 6 Click on the **Families** tab and verify that  $IWaveTheta=0$ , meaning that of all the incident waves, we are going to plot the bistatic pattern of this one only.
- 7 Click **New Report**.  
This generates the report, adding it to the Project tree, and causes it to display.



**8** Click Close to close the Report.

## Creating a Plot for Monostatic RCS

The procedure for creating a Monostatic RCS plot is similar to that used for creating the Bistatic RCS plot.

- 1** Right-click on the Results icon in the Project tree, and select **Create Far Field Report>Rectangular Plot**.

This opens the **New Report - New Trace(s)** dialog.

- 2** For the **Context**, select Monostatic from the menu.
- 3** From **Category** list, select MonostaticRCS.
- 4** From the **Quantity** list, select NormMonostaticRCSTotal (You may need to scroll, or size the dialog to see.)
- 5** From the **Function** list, select dB.

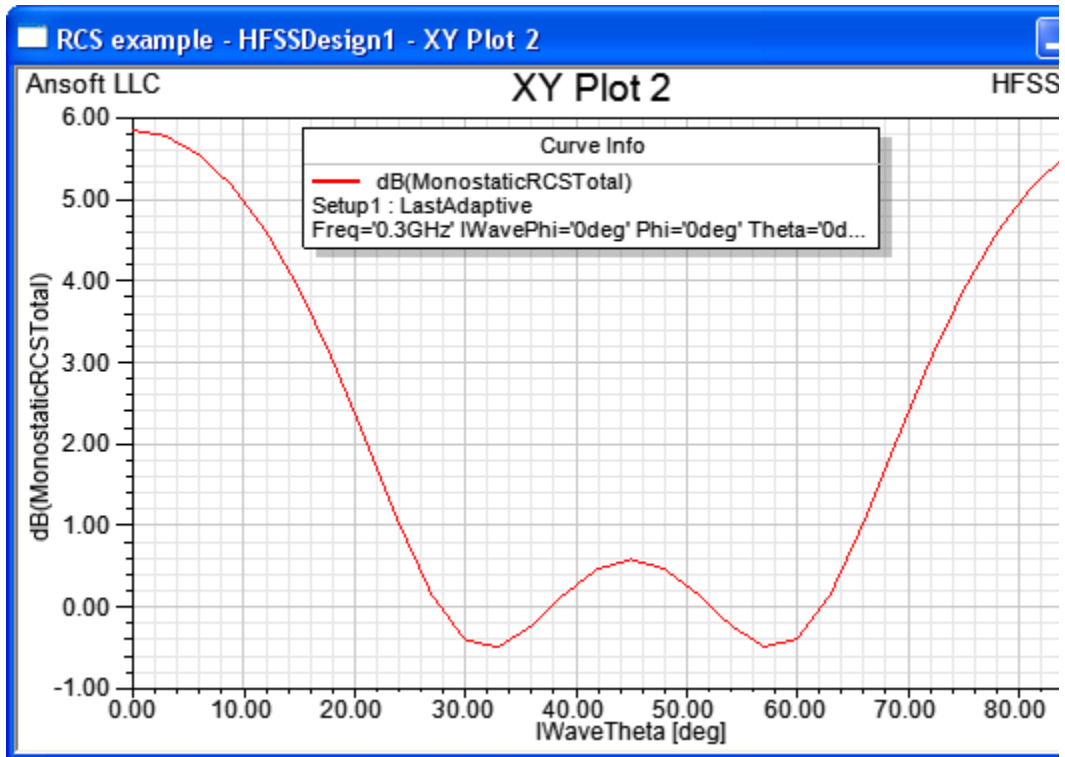
The **Y** field for the Trace should show dB(NormMonostaticRCSTotal).

- 6** Set the **X** field for the Trace as IWaveTheta.
- 7** Click **New Report**.

This generates the report, adding it to the Project tree,



and causes it to display.



This completes the exercise. Save the project and exit.



## 射频和天线设计培训课程推荐

易迪拓培训([www.edatop.com](http://www.edatop.com))由数名来自于研发第一线的资深工程师发起成立,致力并专注于微波、射频、天线设计研发人才的培养;我们于 2006 年整合合并微波 EDA 网([www.mweda.com](http://www.mweda.com)),现已发展成为国内最大的微波射频和天线设计人才培养基地,成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程,广受客户好评;并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书,帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司,以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表: <http://www.edatop.com/peixun/rfe/129.html>



### 射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材;旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习,能够让学员完全达到和胜任一个合格的射频工程师的要求...

课程网址: <http://www.edatop.com/peixun/rfe/110.html>

### ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程,共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解,并多结合设计实例,由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS,迅速提升个人技术能力,把 ADS 真正应用到实际研发工作中去,成为 ADS 设计专家...



课程网址: <http://www.edatop.com/peixun/ads/13.html>



### HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程,是迄今国内最全面、最专业的 HFSS 培训教程套装,可以帮助您从零开始,全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装,更可超值赠送 3 个月免费学习答疑,随时解答您学习过程中遇到的棘手问题,让您的 HFSS 学习更加轻松顺畅...

课程网址: <http://www.edatop.com/peixun/hfss/11.html>

## CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出,是最全面、系统、专业的 CST 微波工作室培训课程套装,所有课程都由经验丰富的专家授课,视频教学,可以帮助您从零开始,全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装,还可超值赠送 3 个月免费学习答疑...

课程网址: <http://www.edatop.com/peixun/cst/24.html>



## HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书,课程从基础讲起,内容由浅入深,理论介绍和实际操作讲解相结合,全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程,可以帮助您快速学习掌握如何使用 HFSS 设计天线,让天线设计不再难...

课程网址: <http://www.edatop.com/peixun/hfss/122.html>

## 13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程,培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合,全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作,同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习,可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试...

详情浏览: <http://www.edatop.com/peixun/antenna/116.html>



### 我们的课程优势:

- ※ 成立于 2004 年,10 多年丰富的行业经验,
- ※ 一直致力并专注于微波射频和天线设计工程师的培养,更了解该行业对人才的要求
- ※ 经验丰富的一线资深工程师讲授,结合实际工程案例,直观、实用、易学

### 联系我们:

- ※ 易迪拓培训官网: <http://www.edatop.com>
- ※ 微波 EDA 网: <http://www.mweda.com>
- ※ 官方淘宝店: <http://shop36920890.taobao.com>