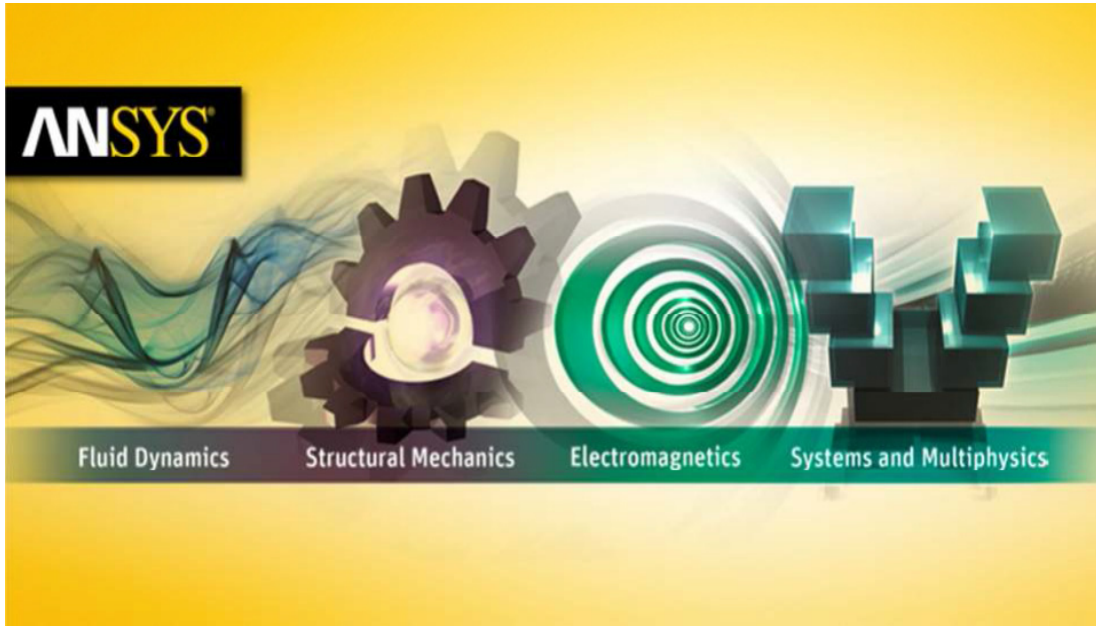

Getting Started with HFSS™

A Ridged Horn Antenna

ANSYS

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

November 2013
ANSYS Electromagnetics Suite 15.0

ANSYS, Inc. is certified to ISO 9001:2008.

Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, HFSS, and Optimetrics 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. 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 SUBSIDIAR IES, 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.

Published in the U.S.A.

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.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Getting Help: ANSYS Technical Support

For information about ANSYS Technical Support, go to the ANSYS corporate Support website, www.ansys.com/Support**.** You can also contact your ANSYS account manager in order to obtain this information.

All ANSYS EM 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**.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

1. Introduction
   - Sample Project: Transient Horn Antenna 1-2
2. Setup the Transient Model
   - Start HFSS and Open the Model 2-2
   - Verify Solution Type 2-2
   - Assign Material to Horn and Pin 2-3
   - Complete the Geometry 2-3
   - Specify Surfaces for Plotting Fields 2-6
   - Boundaries 2-8
   - Assign Excitations 2-9
   - Explanation of Active Ports 2-11
   - Specify a Terminal 2-11
3. Simulation and Results
   - Solution Setup 3-2
   - Simulation 3-6
   - Report 3-10
   - Field Overlay 3-11
   - Radiated Fields 3-14

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, 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 Ridged Horn Antenna. We assume you already have some experience using HFSS in the frequency domain.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sample Project: Transient Horn Antenna

In this project, you will use HFSS Transient to obtain the broad-band S-parameters of a ridged horn antenna. A broad-band antenna such as this can be used to transmit short-duration pulses, e.g. as part of a ground-penetrating radar (GPR) system for the detection of buried pipes or land mines. HFSS Transient is the tool of choice to simulate the interaction of a short-duration electromagnetic signal in the time domain with the antenna and with buried objects.

We will open a model containing the antenna geometry, create the rest of the model, and set up a Transient Network Analysis. We will run the simulation and inspect some results.

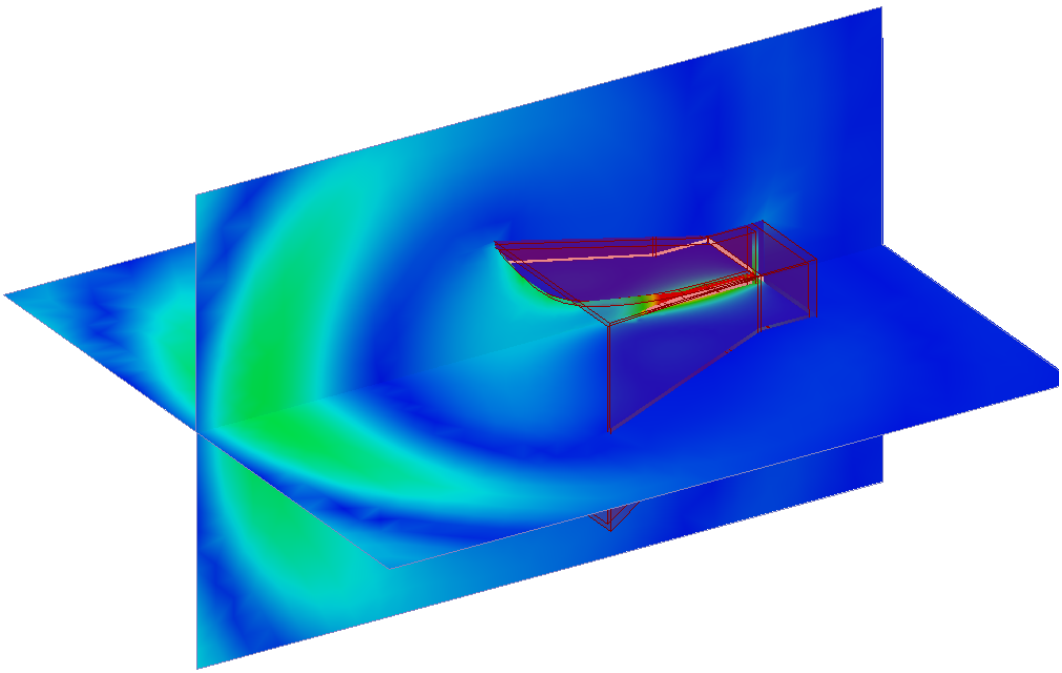

Figure 1. Sample Project

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 Setup the Transient Model

This chapter guides you through the preparation of a model for a transient simulation in HFSS.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start HFSS and Open the Model

We recommend that you store a short-cut of the HFSS application on your desktop.

1. Double-click the HFSS icon to open the application.
2. Use **File>Open** to load the file *broadbandhorn.hfss* from the **Help** folder in the HFSS installation.

   Note Since the project is deliberately incomplete it is located in the **Help** folder as opposed to the **Examples** directory. You will add excitations and a setup before analyzing the model and generating reports.

   Note A sample path could be C:\Program Files\AnsysEM\HFSSn\Win64 or 32\Help.

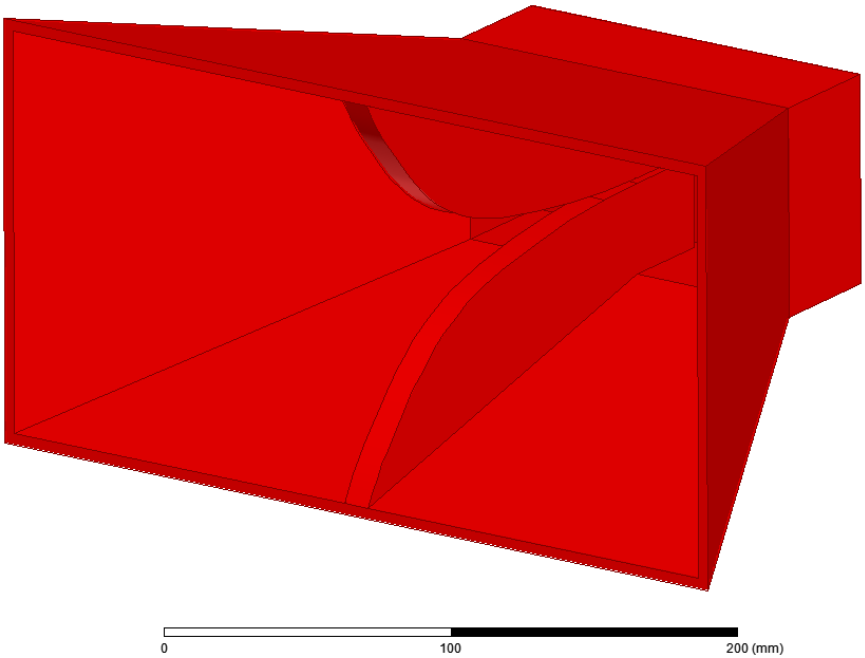

Figure 1. Horn Antenna Model

Verify Solution Type

1. On the **Project Manager** window, right click the option **HFSSDesign*n*(Transient Network)** under **broadbandhorn**

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and select **Solution Type** from the short-cut menu.

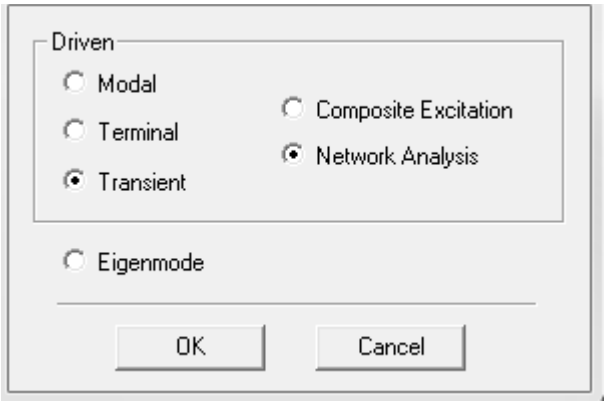

Figure 2. Solution Type

2. Ensure the option **Network Analysis** is selected and click **OK**.
3. Save the project in a different directory of your choice.

Assign Material to Horn and Pin

Notice that the options **horn** and **pin** in the history tree appear under **Not Assigned**.

1. Double-click **horn** to open the **Attribute** dialog box.
2. Set **Material** property as *copper*.
3. Click **OK** to close the **Attribute** dialog box.
4. Double-click pin to open the **Attribute** dialog box.
5. Set **Material** property as *copper*.
6. Click **OK** to close the dialog box.

Complete the Geometry

Complete the geometry by creating an air box and PMLs.

1. Create an air box with starting vertex (X,Y,Z)=(-100, -250, -180) and size (dX, dY, dZ)=(550, 500, 360).

   Note Ensure the fields in the command dialog box are set as shown in the figure below.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

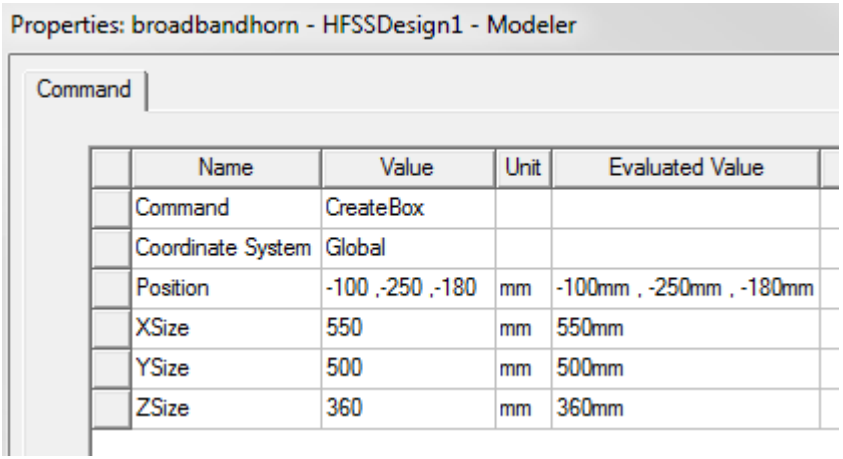

Figure 3. Command dialog box

The faces of this box are at a distance of 100 mm or more from the horn. Since we are going to use PMLs and the minimum frequency will be 0.7 GHz, this is much more strict than necessary, but is still of a reasonable size.

2. Assign air as the **Material**, and enter a name, a color and a transparency of your choice on the **Attribute** dialog box as shown below.

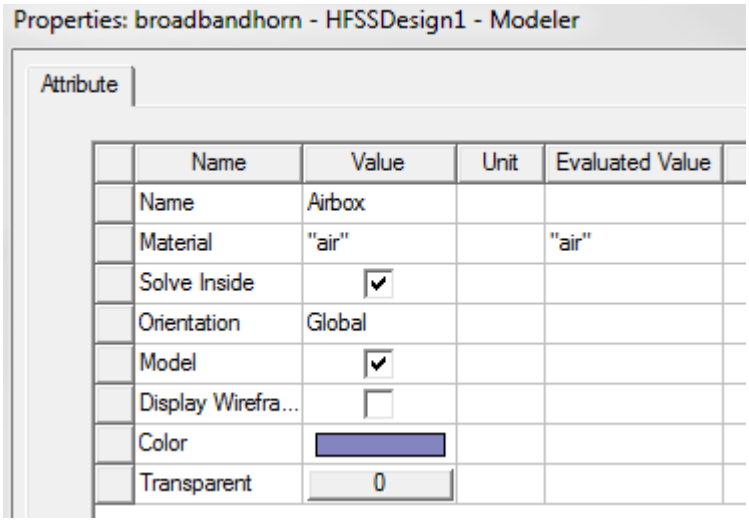

Figure 4. Attribute dialog box

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select all faces of the air box.
4. On the toolbar click **HFSS>Boundaries>PML Setup Wizard**.
5. Edit the thickness: 100 mm. See Figure 5 below.

   Note The thickness set is somewhat arbitrarily, since the wizard will adjust the material parameters to obtain good absorption at a given thickness.

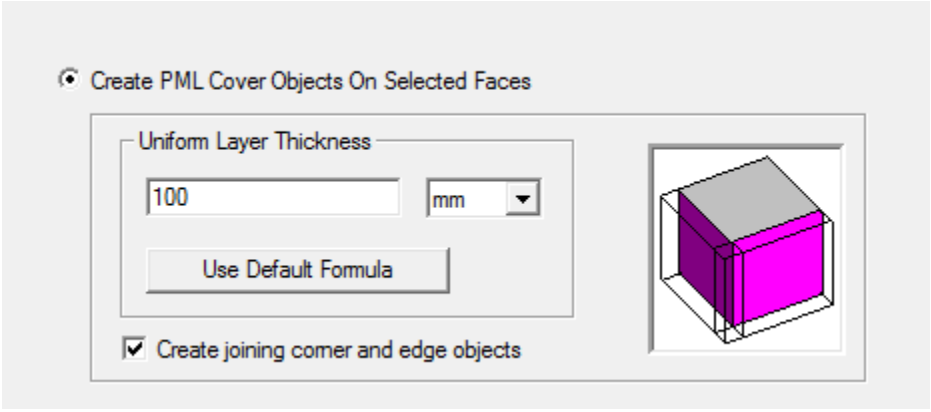

Figure 5. Specifying the thickness

6. Click **Next**.
7. Set the fields in the PML Setup Wizard: Material Parameters dialog as shown in Figure 6.

   Note These specifications are needed to enable HFSS to generate the correct absorbing materials, materials that absorb very well at the given layer thickness, frequency and distance from the nearest part of the radiating structure, and that absorb even better at higher frequencies and larger distances.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

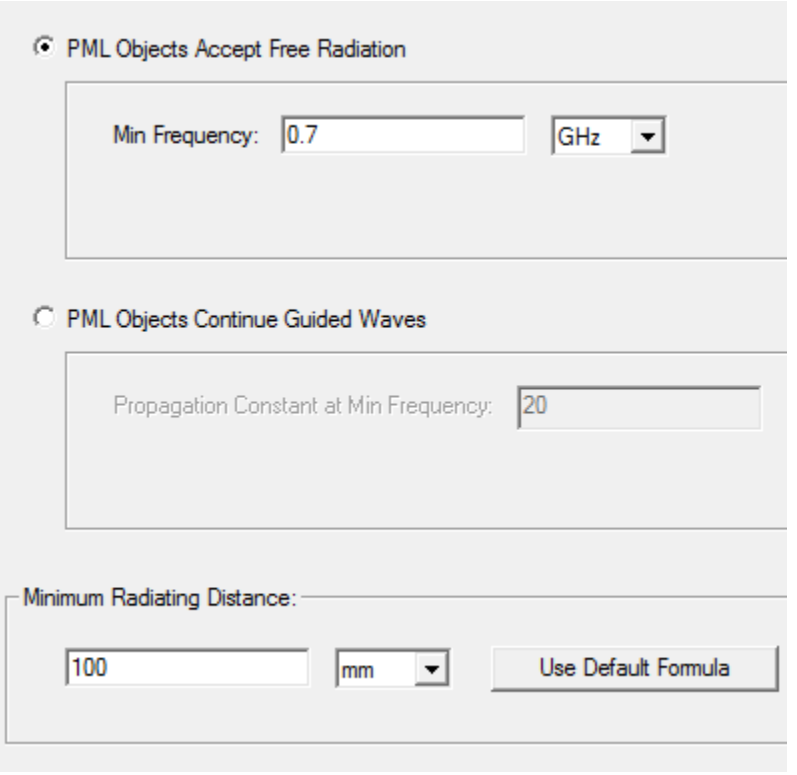

Figure 6. PML Setup Wizard: Material Parameters

8 Click **Next** and then, click **Finish**.

Specify Surfaces for Plotting Fields

We need to decide at an early stage where we want to visualize fields, since saving fields in the entire 3D model at every time step is not practical. We will create two rectangles where we want to save fields.

1 Create a rectangle in the XY plane starting at (X,Y,Z)= (-100, -250, 0) with size (Xsize, Ysize)=(550, 500).

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate...	Global		
Position	-100 ,-250 ,0	mm	-100mm , -250mm , 0mm
Axis	Z		
XSize	550	mm	550mm
YSize	500	mm	500mm

Figure 7. Command dialog

2 Create a rectangle in the XZ plane starting at (X,Y,Z)= (-100, 0, -180) with size (Xsize, Zsize)=(550, 360).

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate...	Global		
Position	-100 ,0 ,-180	mm	-100mm , 0mm , -180mm
Axis	Y		
XSize	550	mm	550mm
ZSize	360	mm	360mm

Figure 8. Command dialog

3 Display both rectangles as wire frames.

We need to define a “list” in order to specify later that we wish to save fields here.

4 Select both rectangles.

5 **Modeler>List>Create>Object List** (or **Face List** if you selected them in face-select mode)

6 Name the list as “PlotFields”, as shown below, through the **Model Tree** and the **Properties**.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

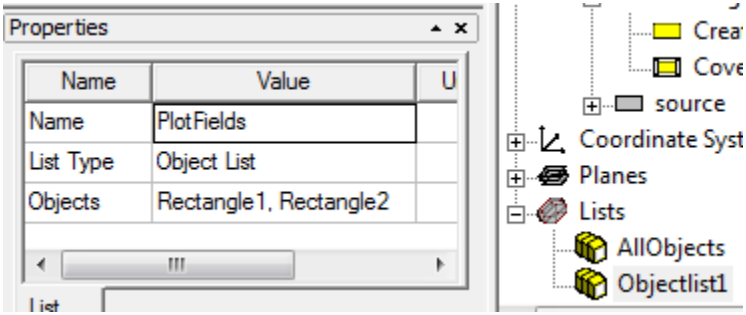

Figure 9. Changing the name “Objectlist1” to “PlotFields”

7 Save the model.

Boundaries

PML Radiation faces were defined when the PMLs were created. Expand **Boundaries** on the project tree to see the list.

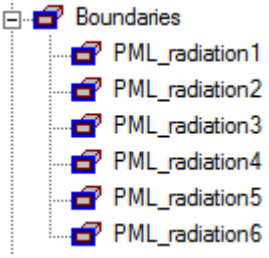

Figure 10. PML boundaries list

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Excitations

The sheet object “source” will be a wave port.

1. Click **Tools>Options>HFSS Options** to see the **HFSS Options** dialog box.

   Under the General tab, under Assignment Options see that you have unchecked Auto-assign terminals on ports, and click **OK**.

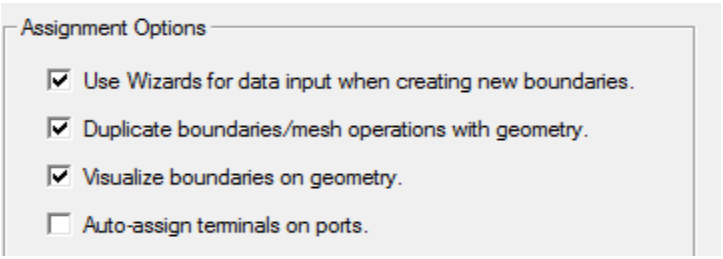

Figure 11. Assignment Options

2. To locate the source, select it from the history tree and then, use the option **Drag rectangle using left mouse to zoom in the camera.**

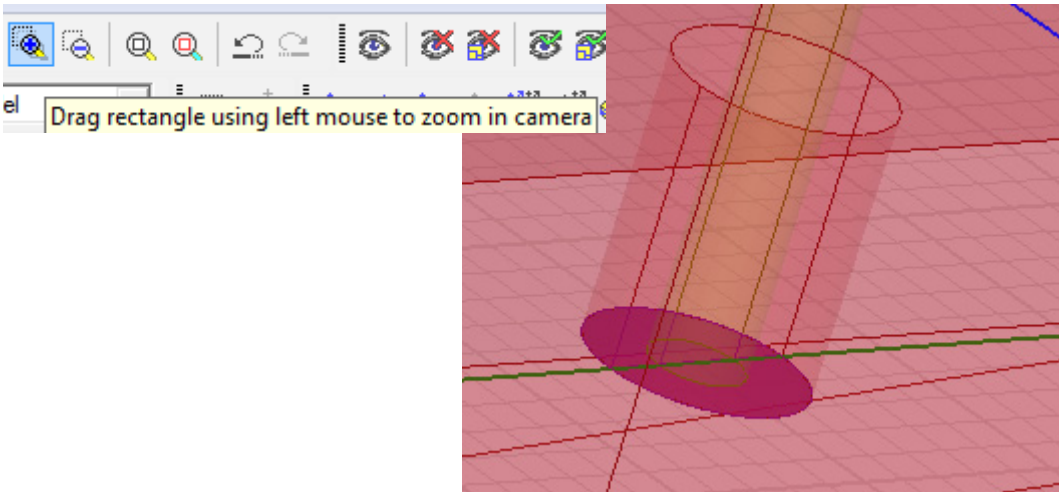

Figure 12. zoom-in to see the source

Note Notice how this object can serve as the source of a

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

co-axial cable of which the outer conductor is part of one ridge and the inner conductor connects to the opposite ridge.

3. **Assign a Wave Port** to this sheet object.
4. Give the name as p1 and click **Next**.
5. Specify that the port will be **Active. Click Next.**

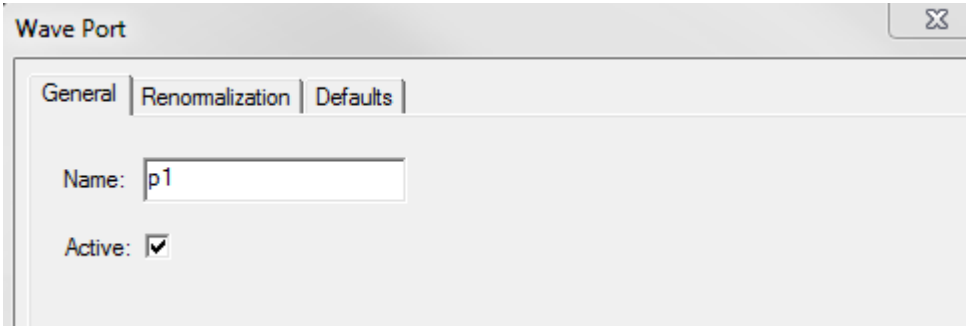

Figure 13. Specify port to be active.

6. Select **Do Not Renormalize**.

   The port impedance is expected to be close to 50 Ohm, and seeing the actual impedance can confirm that the model has been set up correctly.

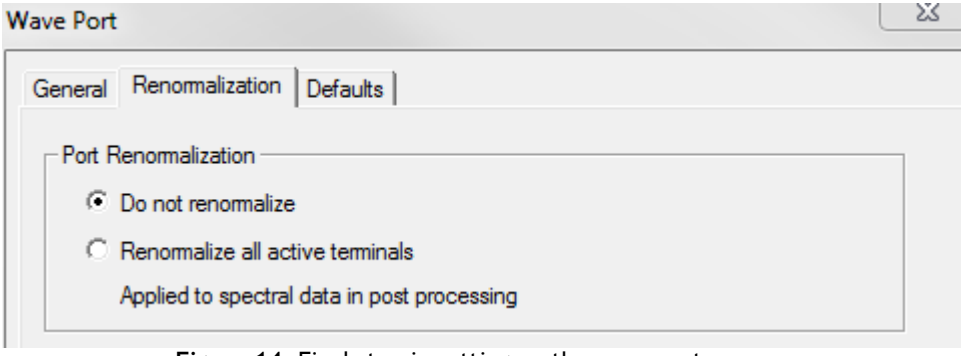

Figure 14. Final step in setting up the wave port

7. Click **Finish**.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explanation of Active Ports

In a Transient Network Analysis design with multiple ports, in order to obtain the full S-matrix, the ports will be ON one at a time for a full 3D simulation. Each simulation with a particular port ON will produce one column of the S matrix. When you specify a port as passive, it will only act as termination. This saves simulation time, at the expense of not generating the particular column of S-parameters.

In a general Transient (non network analysis) design, all active ports will be ON simultaneously, while the remaining passive ports will act as terminations only.

In both cases, the distinction between active and passive ports gives you flexibility.

Specify a Terminal

Since HFSS Transient uses Terminal-Driven ports, we need to specify a terminal.

1. Right-click p1 in the project tree and select **Auto Assign Terminals** from the short-cut menu.

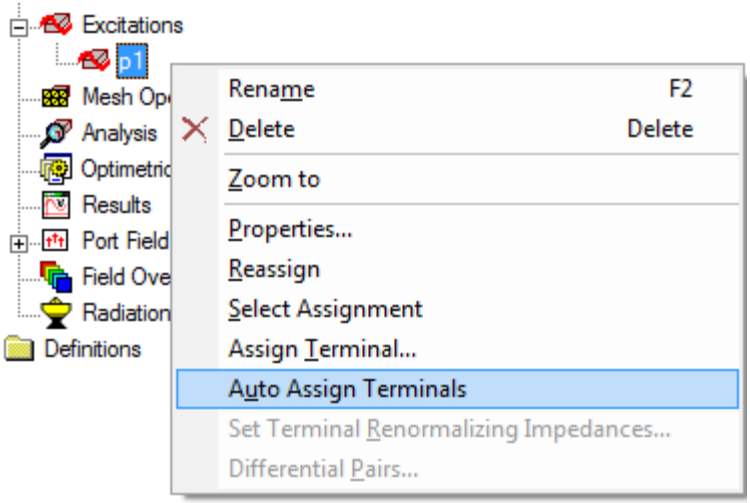

Figure 15. Auto-assigning terminals once the port has been defined

2. Specify the horn as **Reference**, so the pin will be the ter-

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

minal.

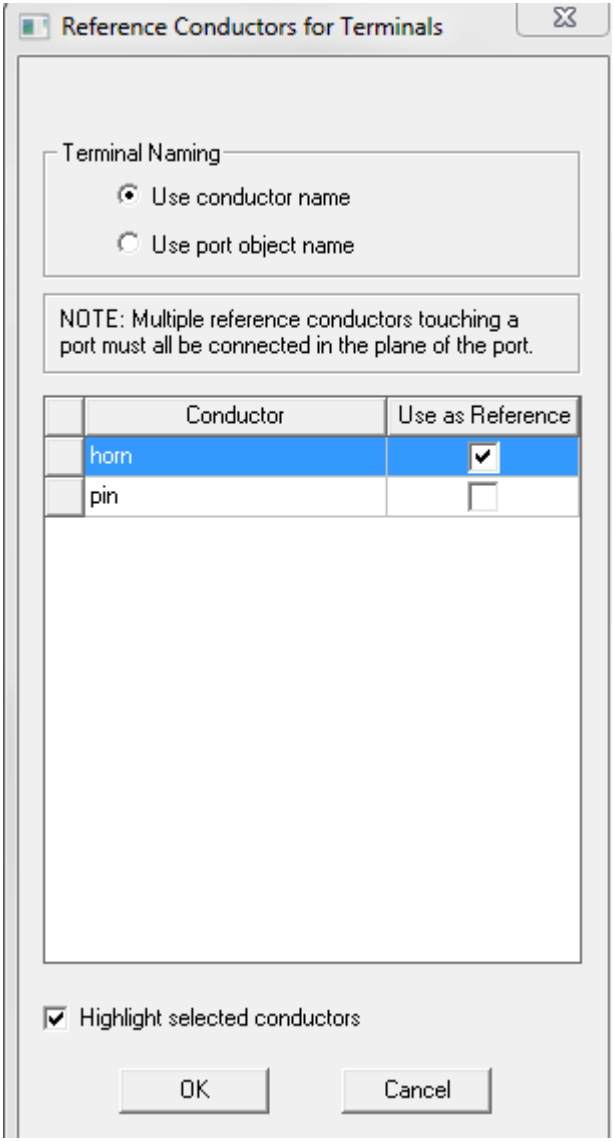

Figure 16. Specify the conductor touching the port to be the reference

Zoom in to see the terminal associated with the wave port.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

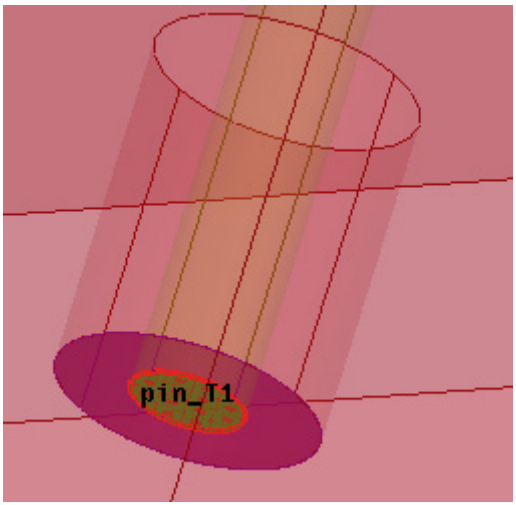

Figure 17. terminal

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 Simulation and Results

This chapter describes how to setup the transient simulation for the horn antenna and view the results.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Setup

1. In the Project Tree, select **Analysis>Add Solution Setup**.
2. The mesh for the transient simulation is generated by a regular frequency-domain simulation. For that simulation, the software will decide on the appropriate frequency at which to perform the adaptive passes. It will use mixed element orders and the iterative solver. Accept the defaults under the **General** tab.

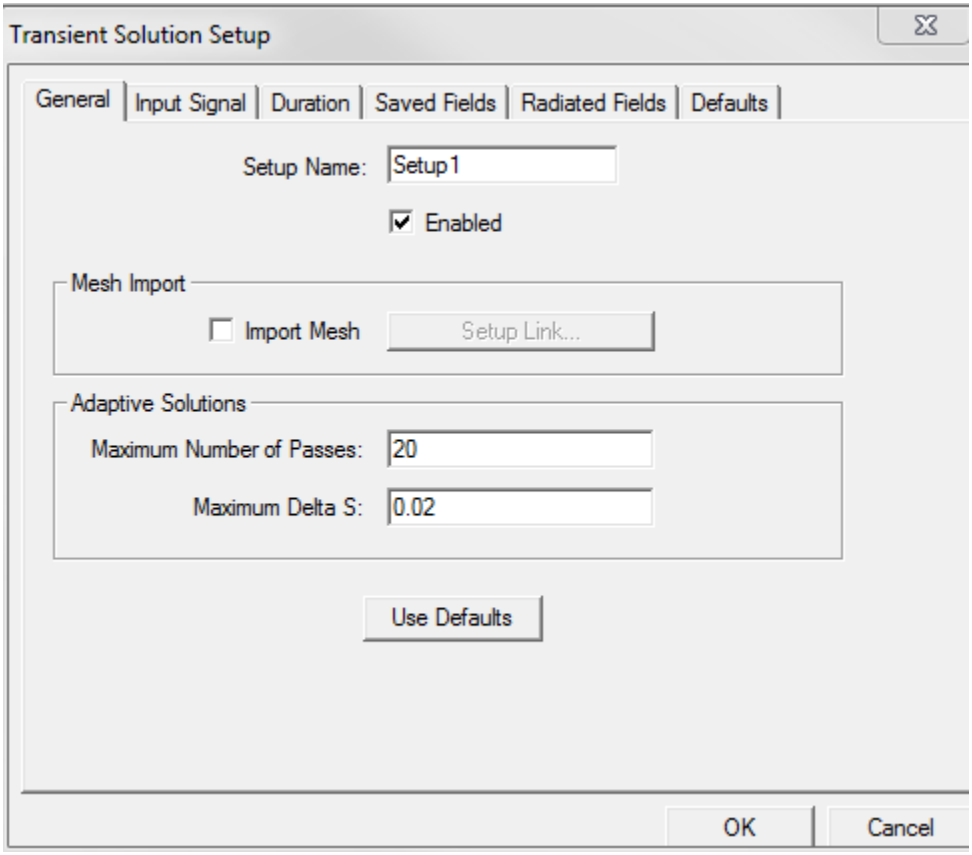

Figure 1. General tab

3. Under the **Input Signal** tab, specify the frequency band of interest as shown below. This band is covered by a modu-

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lated Gaussian pulse in the time domain, as can be seen in the panel.

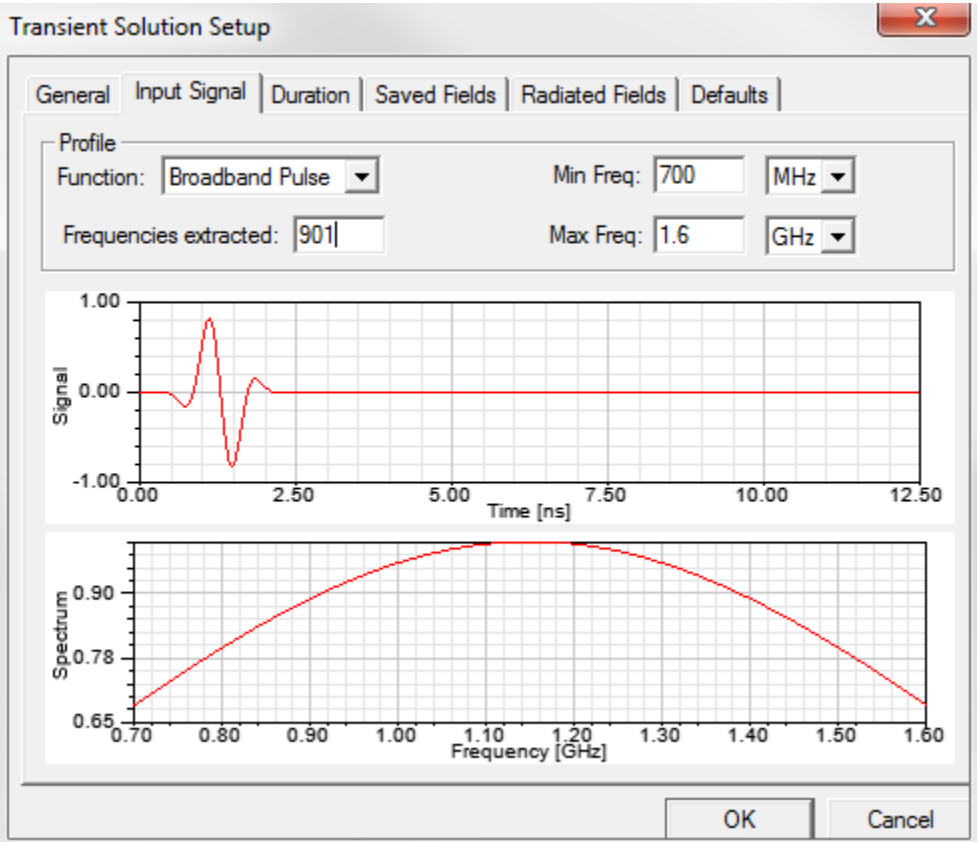

Figure 2. Input Signal

4 Under the **Duration** tab, you specify limits for the simulated time of the transient analysis. For this exercise, select **Auto Terminate**.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

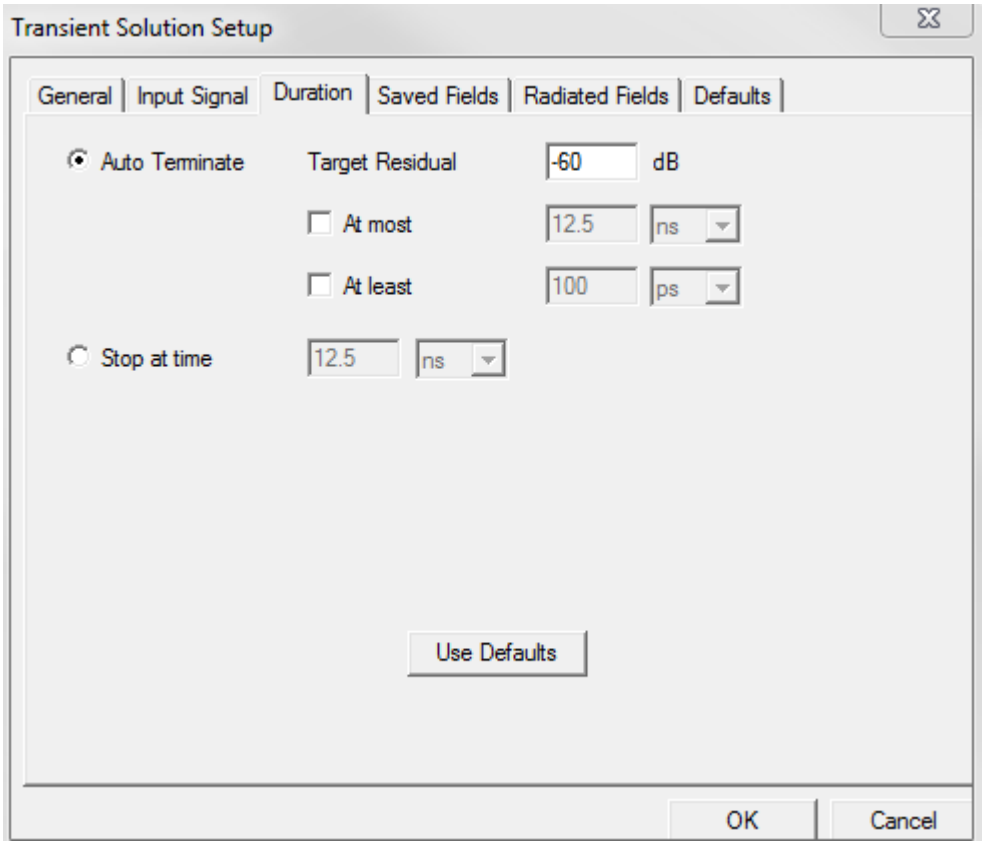

Figure 3. Duration tab

The Target Residual is based on the maximum field remaining in the model at a given time, relative to the all-time high. Once the fields have fallen below 0.001 of the all-time high, the simulation can safely be stopped. Additionally, you can specify a maximum and a minimum time span to be simulated. The built-in maximum and minimum time spans are appropriate in most simulations. They take the model size and the type of signal into account. Only in special cases you may find that you need to override the defaults.

5 Under the **Saved Fields** tab, check the Object List or Face List you defined earlier. Ask for fields to be saved every 30

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ps. This is a small-enough fraction of the duration of the broadband pulse to obtain a smooth field animation later on.

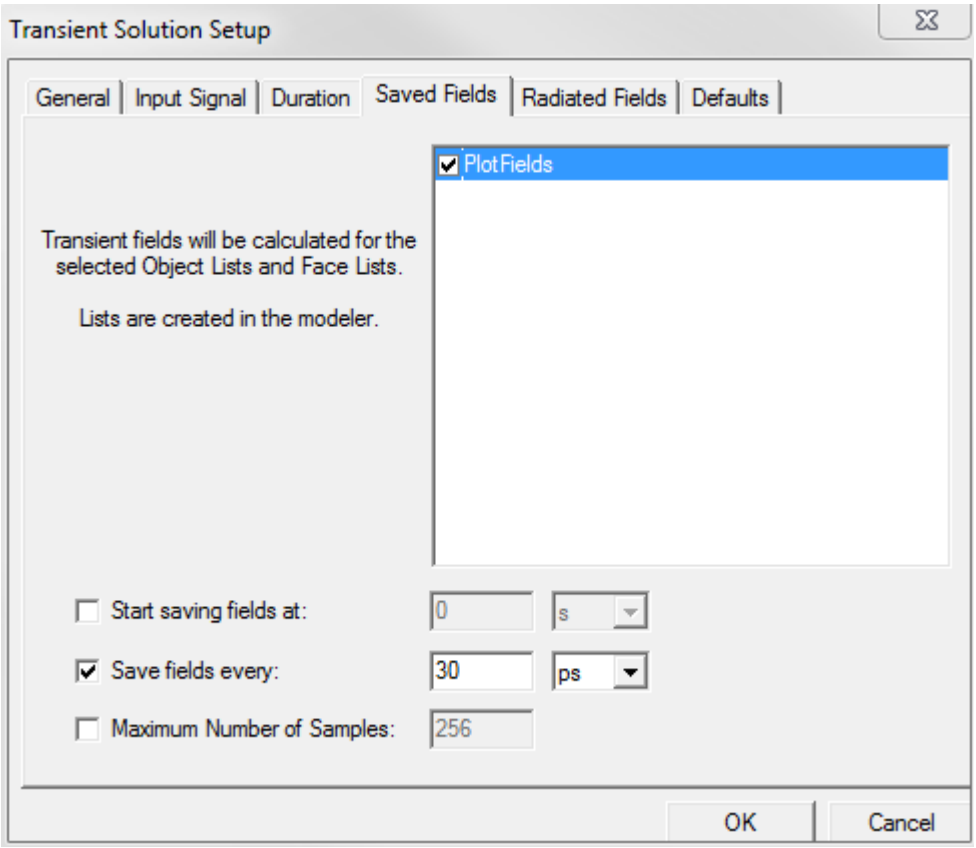

Figure 4. Saved Fields

6 Under the **Radiated Fields** tab, ask for both time-domain radiated fields and frequency-domain radiated fields at 1.2 GHz, as shown below.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

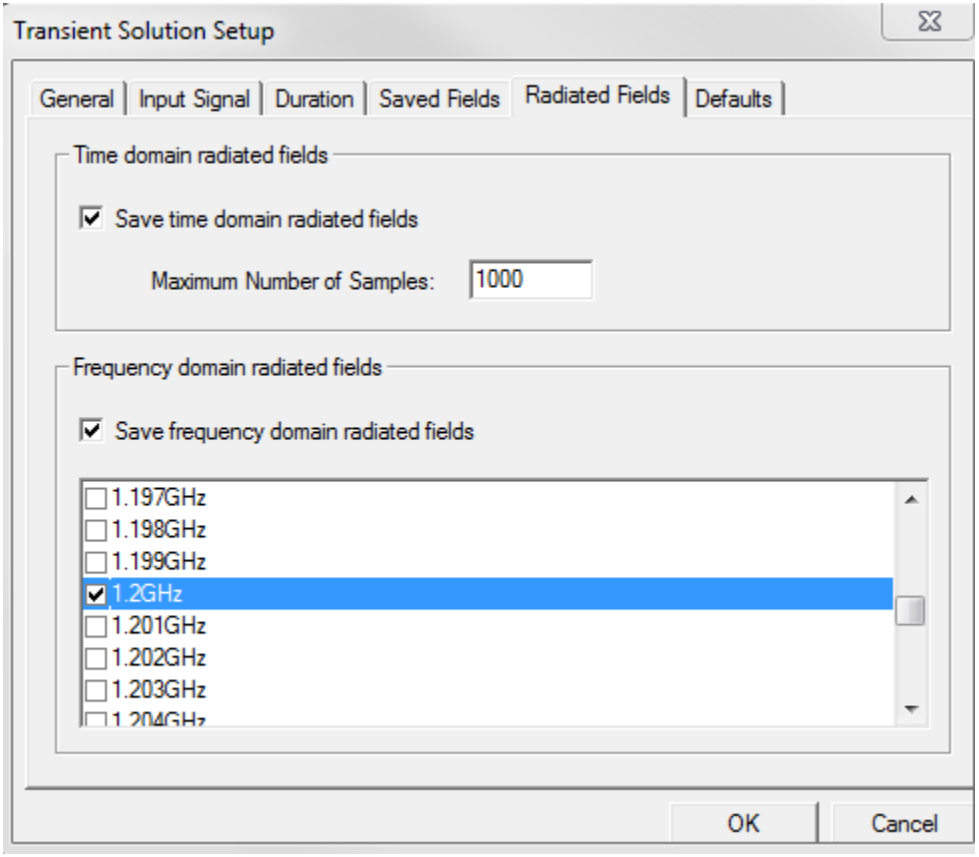

Figure 5. Radiated Fields

7 The **Solution Setup** is complete. Click **OK**.

Simulation

1 Run the simulation.

The progress bar is based on the maximum simulated time from the Duration tab in the Solution Setup, or, if none is specified, on 20×(model diagonal)/(speed of light). In general, the simulation is done well before the progress bar has reached its maximum length.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

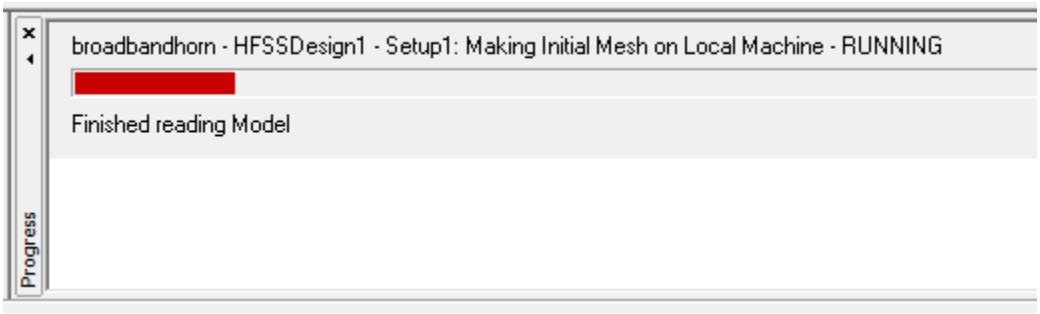

Figure 6. Progress bar, displaying simulated time relative to its maximum

While the simulation is running, you can follow its progress in other ways.

2 Right-click on **Results** in the project tree and select **Create Terminal Solution Data Report>Rectangular Plot**.

Plot the quantities **Input** and **Output** in one report. Plot **Residual** in dB20 in another.

Input and Output show the excitation and the reflection in the port as a function of time.

Residual shows a measure for the maximum field level in the model. The simulation is complete when Residual falls below 0.001 of its all time peak. Change the setting to Volts from mV to view the results.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

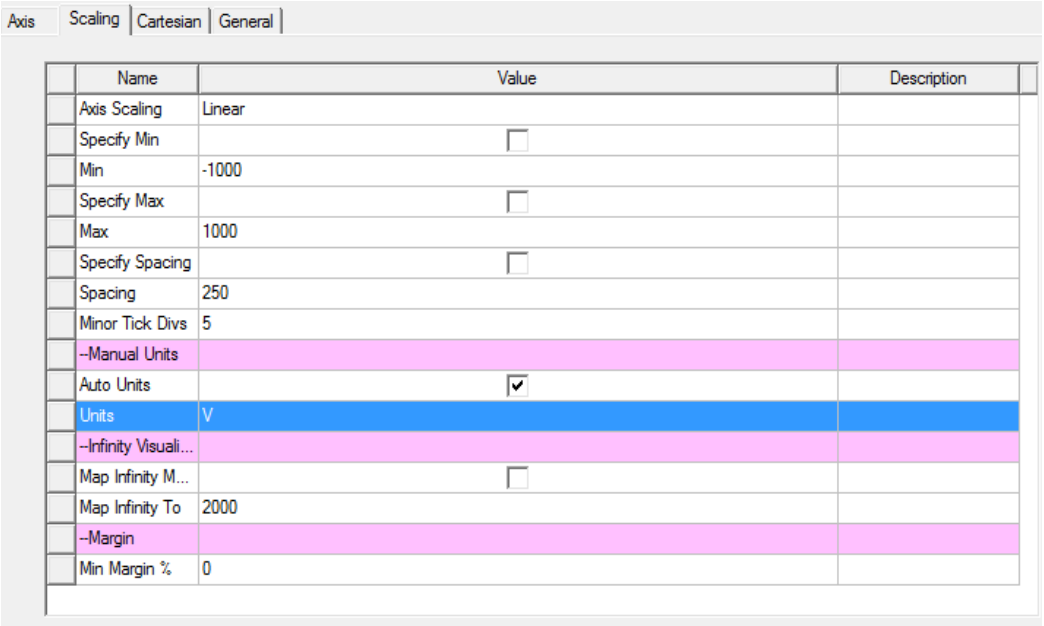

Figure 7. Modify from mV to Volts

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

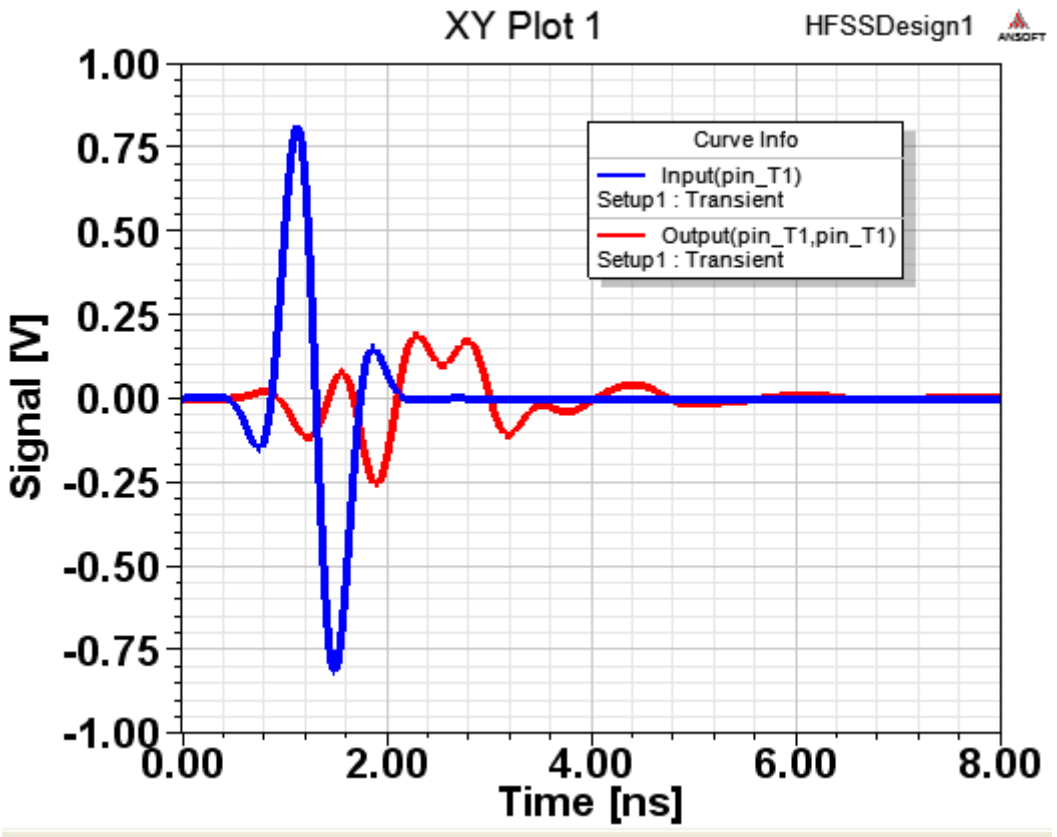

Figure 8. Input and Output in volts (excitation and reflection) as a function of time

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

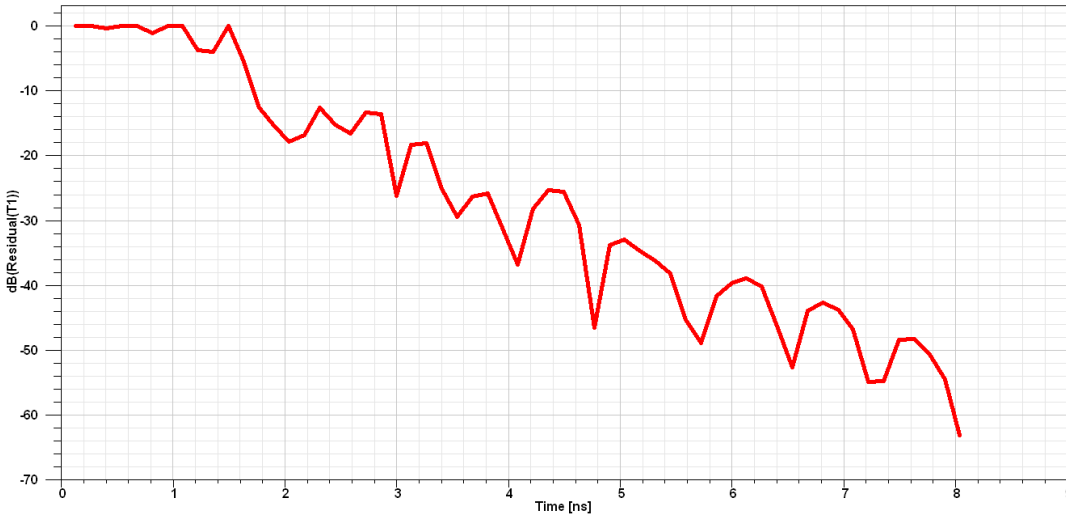

Figure 9. Residual on a log scale, as a function of time

Report

1. To plot S-parameters as a function of frequency, again right-click on **Results** in the project tree and select **Create Terminal Solution Data Report>Rectangular Plot**.
2. Change the Solution to Spectral as shown below.

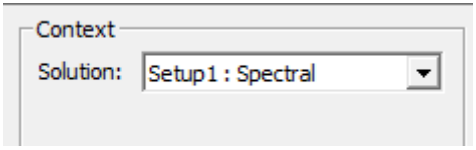

Figure 10. Preparing a report of S-parameters as a function of frequency, in the Spectral Solution

3. Plot the magnitude of S_{11}.
   The resulting report is shown below. This report can also be generated while the simulation is still running. That enables you to see at an early stage where the results are

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

headed.

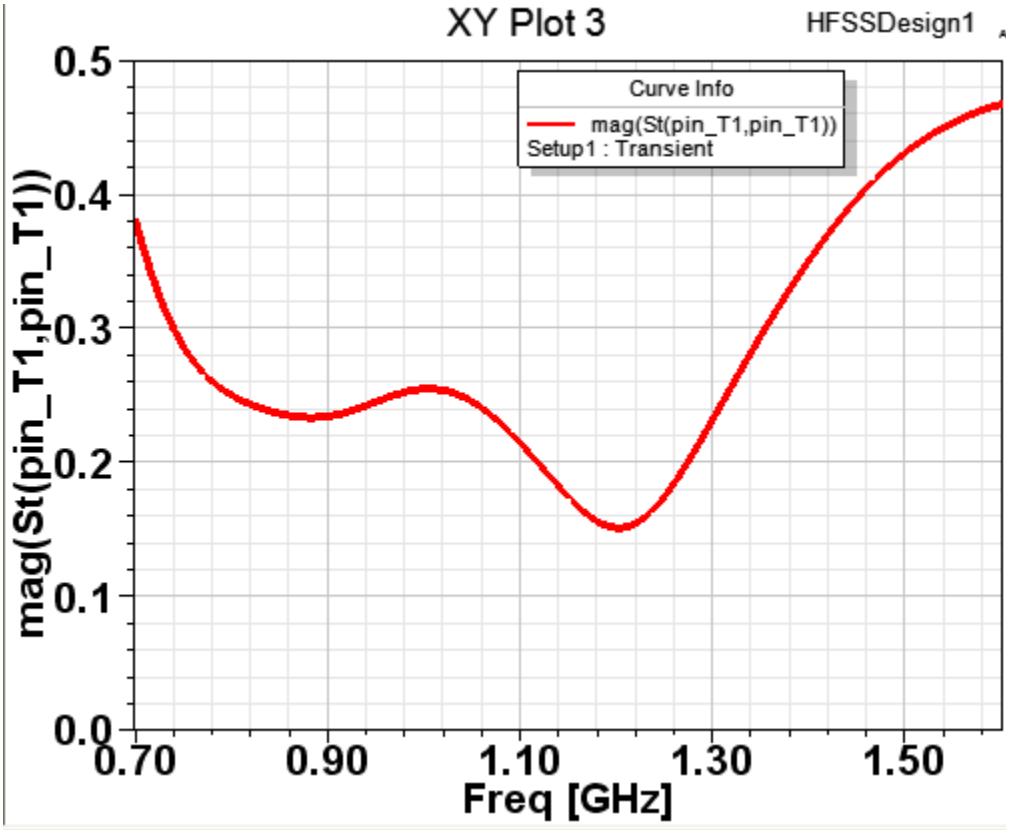

Figure 11.
$|S_{11}|$ *for the broad-band horn*

Field Overlay

1. To create a field overlay, **select** the list named **"plot-fields"** in the model tree, i.e. the list containing the two rectangles you had created for the purpose of field plotting.
2. In the Project Tree, right-click **Field Overlays>Plot Fields>E_t>Mag_E_t** and select an arbitrary nonzero time in the next panel. You can always adjust the time later by right-clicking the plot name and selecting **Modify.**

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

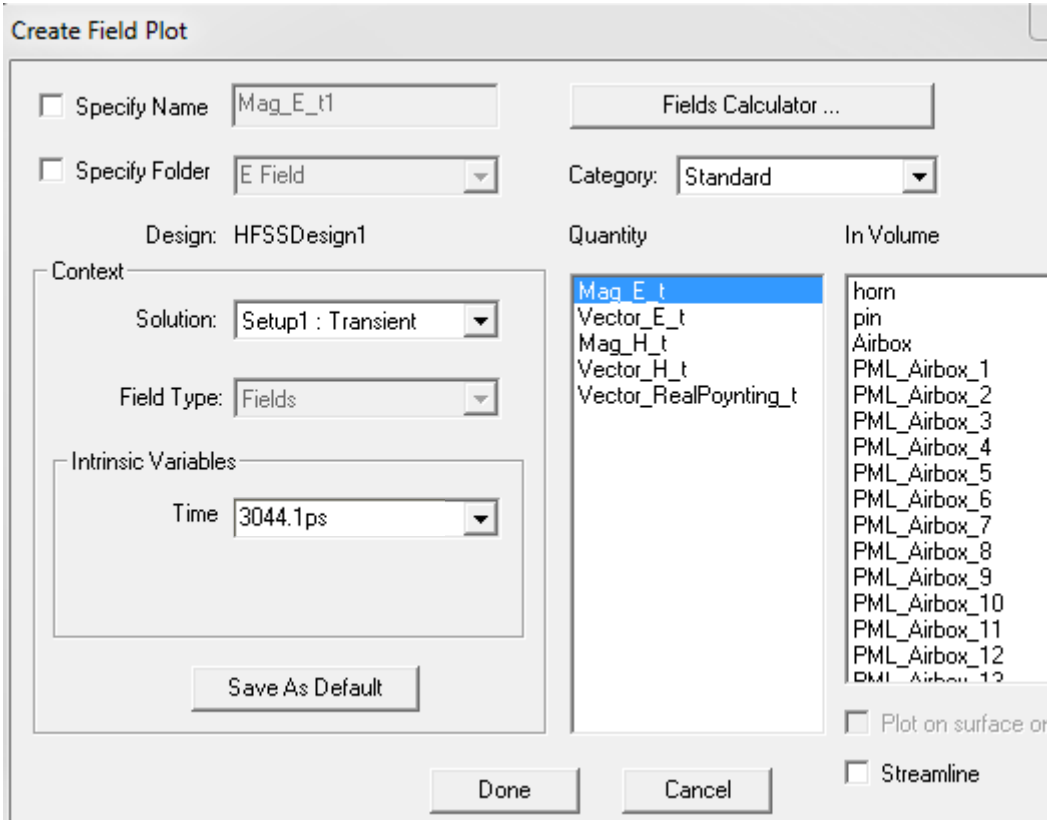

Figure 12. Field Overlay at 3.044ps

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The field overlay is created.

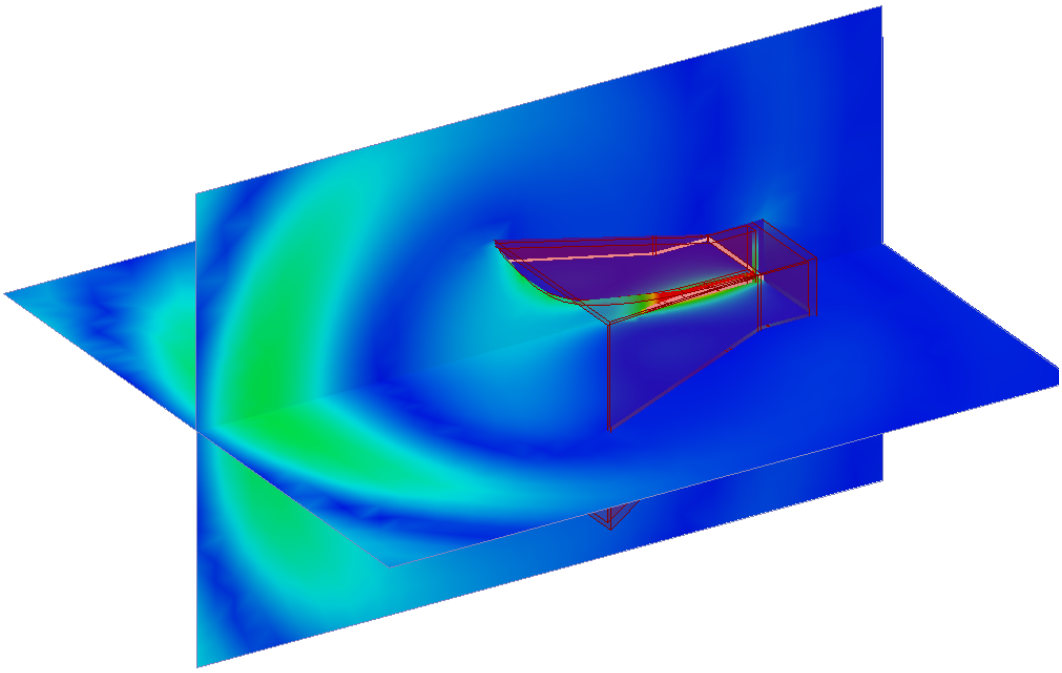

Figure 13. Example field overlay

3. Change the scale to make it range from 0 to 10 V/m.
4. Animate the plot by right-clicking on the plot name and selecting **Animate**.

   In the **Setup Animation** window, you can adjust the number of steps. If necessary, HFSS will interpolate between saved field solutions.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

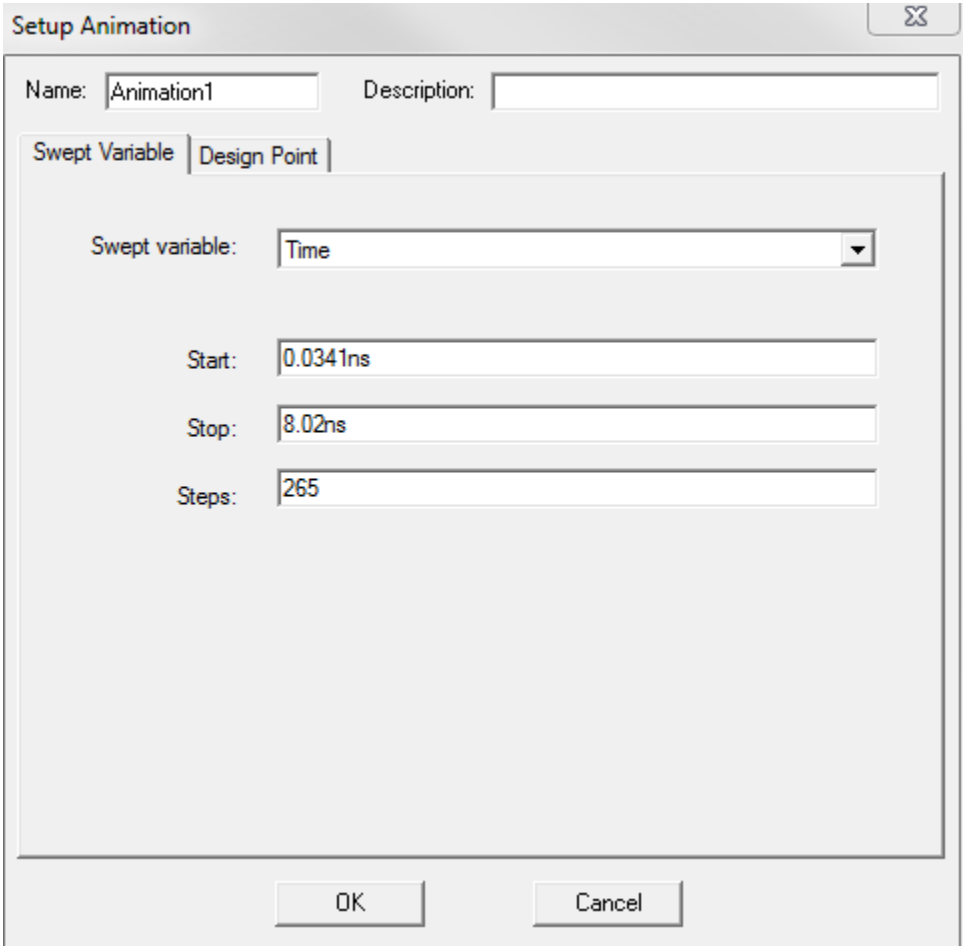

Figure 14. Setup Animation window

Radiated Fields

1 To compute transient radiated fields, right-click on **Radiation** in the Project tree and select **Insert Far Field**

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup>Infinite Sphere

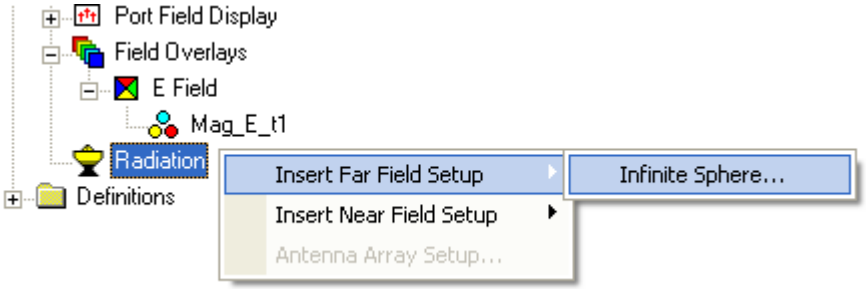

Figure 15. Inserting a far-field setup

2 In the Project Manager, right-click on Radiation and select **Insert Far-Field Setup>Infinite Sphere**. Specify a range of interest as shown. This range is in the XZ plane, and spans from zenith to nadir in the “forward” direction of the antenna

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

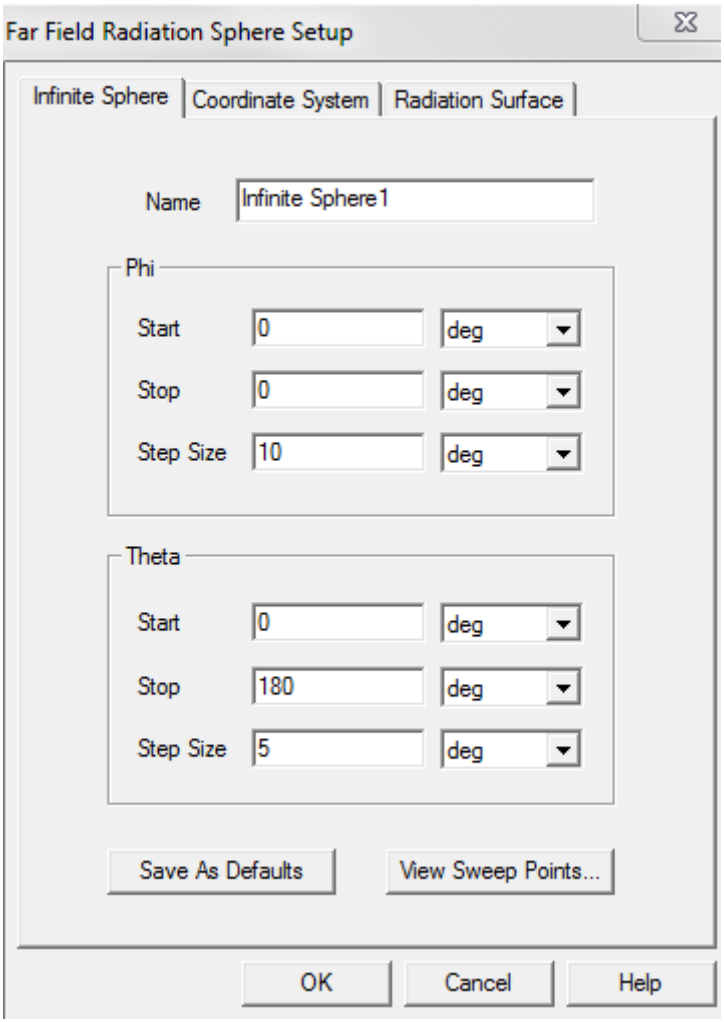

Figure 16. Specifying only one point on the far-field sphere

3. Next, in the Project tree, right-click **Results>Create Far-Fields Report>Rectangular Plot**
4. Create a report of $\mathbf{rE_z}$ as a function of time.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

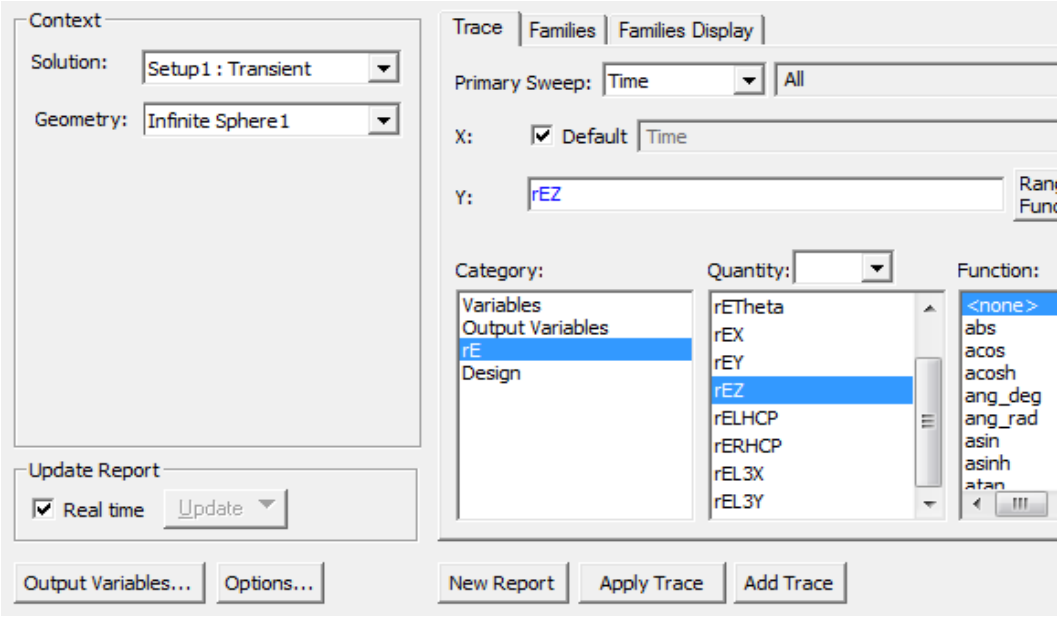

Figure 17. reports panel

5 Under the Families tab, specify Theta=90 deg and phi=0 deg, the center of the main beam.

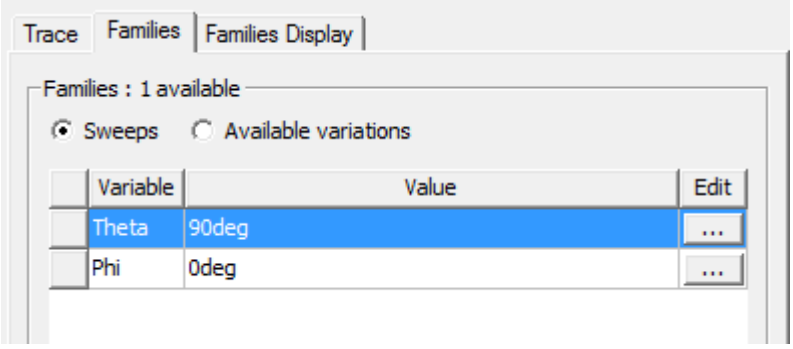

Figure 18. Families tab

Explanation: E_z is the dominant, co-polarized, component of E. The product of r (distance) and E_z is practical because it's independent of distance in the far field. The unit of rE is Volt, since the unit for E is V/m which is multi-

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plied, through r, by meters. The question is sometimes asked which distance HFSS uses in this calculation. The distance is infinite. The expression for far radiated fields has the form E=U/r where U contains an integral over radiation surfaces and r approaches infinity. Multiplying both sides by r yields an expression for rE at infinity without the need to evaluate the radiated E at a finite distance.

The figure below shows the shape of the far radiated field along the main beam. A plot like this is instrumental in determining how the original excitation is distorted by the antenna. Information like this can be used to improve the antenna design, e.g. by judiciously placing resistive strips in certain locations to suppress late-time ringing. It is also useful information in the development of signal-processing algorithms.

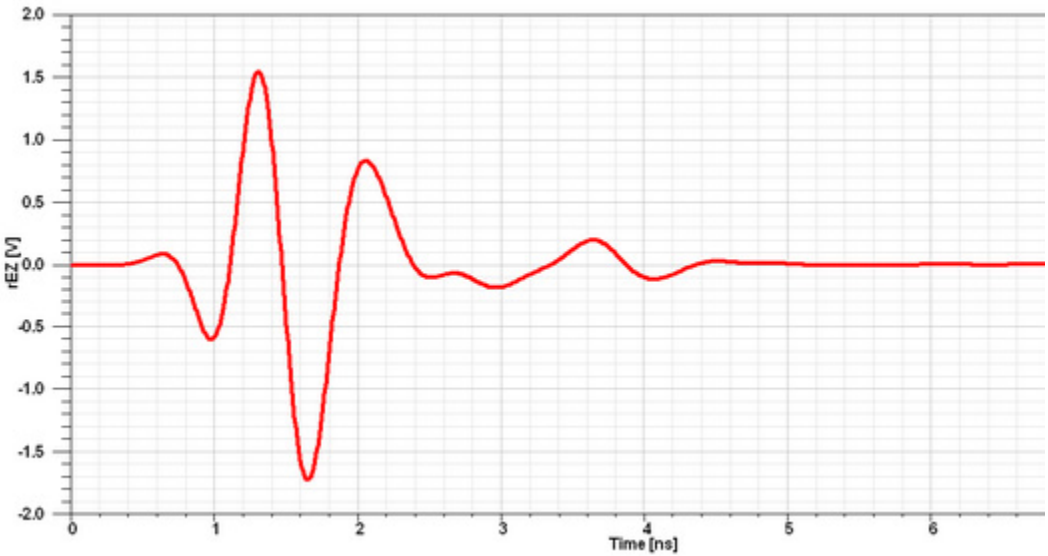

Figure 19. Far radiated field, co-polarized, in the center of the main beam.

Optionally, you can also create a frequency-domain far field by changing the solution to spectral and changing the primary sweep to Theta, as shown below.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

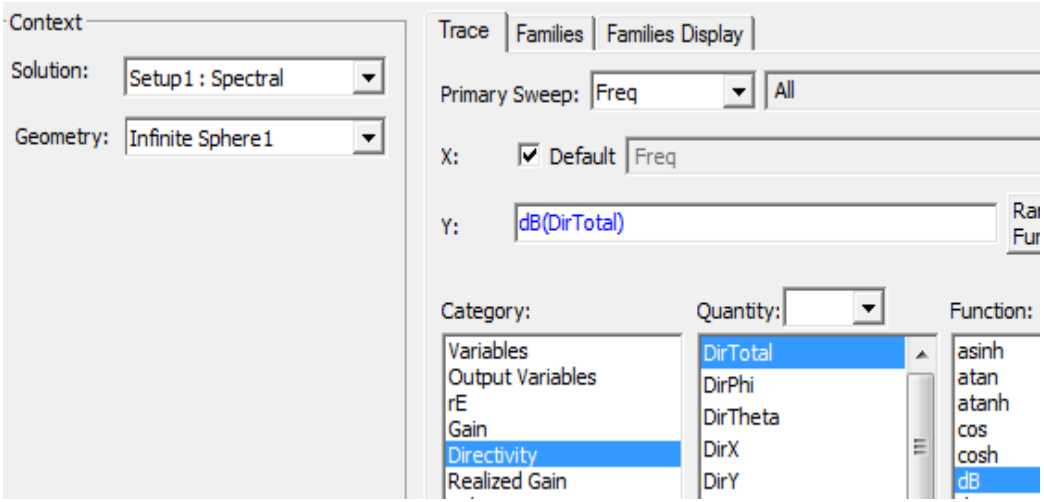

Figure 20. panel

Under the families tab, you will notice that the frequency of 1.2 GHz, specified earlier under the Radiated-Fields tab of the Solution Setup, is automatically available.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Electromagnetics Suite 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.