Ansys 2024 R2

Take A Leap Of Certainty™

Getting Started with HFSS: Ridged Horn Antenna

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

Release 2024 R2
July 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Conventions Used in this Guide

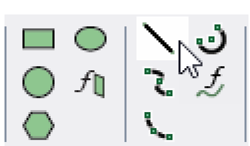
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). 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** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- 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, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

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

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Introduction

This document is intended as supplementary learning material for HFSS. It is suitable for beginners through advanced users. This guide includes instructions to set up a *transient* analysis, solve, and evaluate the results of a double-ridged broadband horn antenna. We assume you already have some experience using HFSS in the frequency domain.

Sample Project: Broadband Ridged Horn Antenna

A broadband antenna, such as the one in this project, can be used to transmit short-duration pulses. Application examples include ground-penetrating radar (GPR) systems for the detection of buried pipes or land mines. HFSS Transient is the tool of choice to simulate the interaction, in the time domain, of a short-duration electromagnetic signal between the antenna and buried objects.

For this problem, perform an HFSS Transient analysis to obtain the broadband S-parameters, E-fields overlay, and radiated fields of the ridged horn antenna.

You will open a partially completed model containing the antenna geometry, complete the model, and set up a Transient Network Analysis. You will then run the simulation and evaluate the previously mentioned S-parameter and field results.

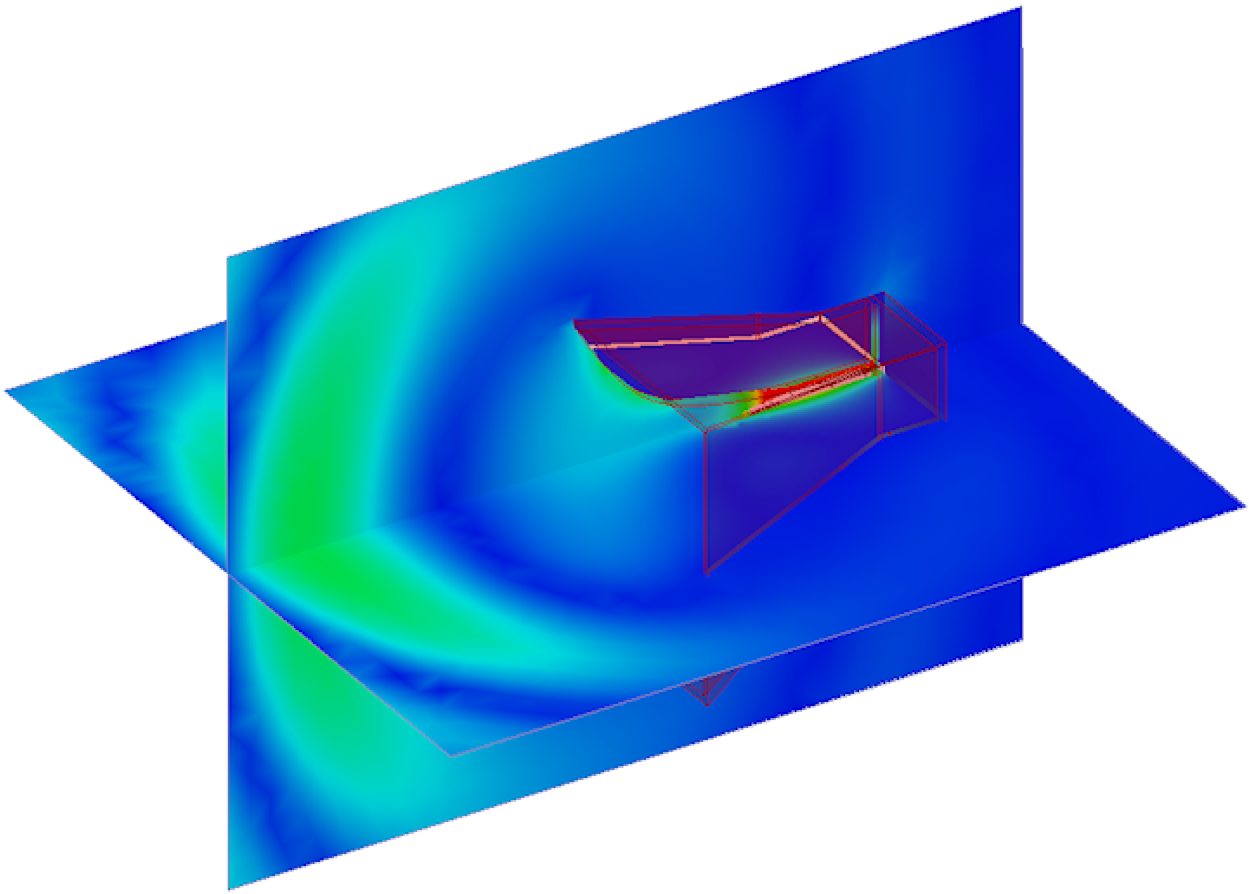


Figure 1-1: Sample Project

2 - Set Up the Transient Model

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

Open the Model

Begin with the Ansys Electronics Desktop application launched but with no project open.

1. On the **Desktop** ribbon tab, click **Open Examples**. Then:
   a. In the *Open* dialog box that appears, click the parent folder icon ( ) once to move up one level above the *Examples* folder.
   b. Double-click the **Help** folder and then the **HFSS** folder.
   c. Select the file **broadbandhorn.aedt** and click **Open**.

   The model appears in the Modeler window:

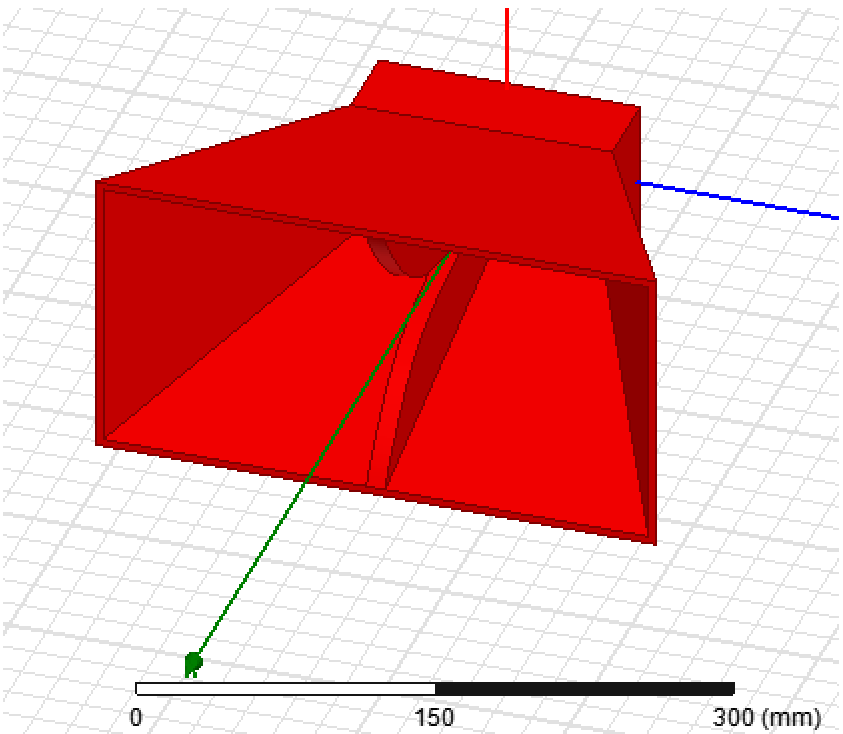


Figure 2-1: Ridged Horn Antenna Model

> **Note:**
>
> This project is deliberately incomplete. You need to assign materials and excitations, create an open region, designate faces where fields are to be saved, setup the solution, analyze the model, and generate the reports and field overlay. For this reason, the project file is located in the *Help* folder instead of the *Examples* folder.

2. On the **Desktop** ribbon tab, click  **Save As**.
3. Navigate to a working folder of your choice (you can't write to the program installation path) and click **Save** to place a copy of the model in your working folder using the same file name.

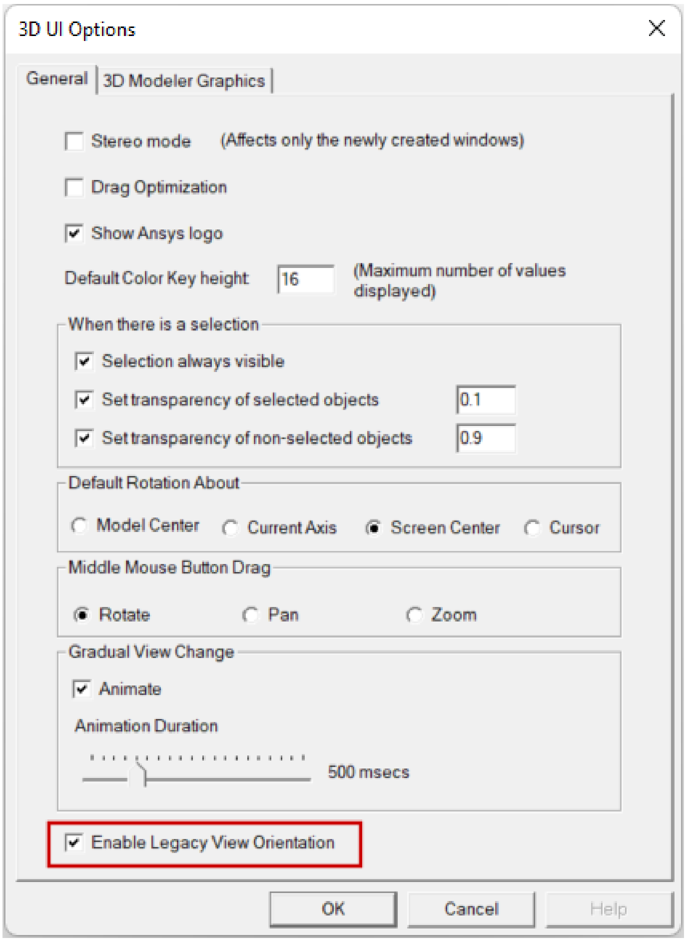
Enable Legacy View Orientations

This getting started guide was created based on standard view orientations that were in effect for version 2023 R2 and earlier of the Ansys Electronics Desktop application. For consistency between your experience and the views and instructions contained in this guide, select the *Enable Legacy View Orientation* option in the 3D UI Options dialog box, as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Select **Enable Legacy View Orientation**:



3. Click **OK**.

   Changing the view orientation option does not change the model viewpoint that was in effect at the time.

4. On the **Draw** ribbon tab, click **Orient** to change to the *Trimetric* view, which is the default legacy view orientation.

   You do not have to select *Trimetric* from the *Orient* drop-down menu. The default view appears when you click *Orient*.

Although this option can only be accessed once a design is added to a project, it is a global option. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

At the end of this guide, you will be prompted to clear the *Enable Legacy View Orientation* option, if you prefer to use the view orientation scheme implemented for 2024 R1 and newer versions going forward.

For a comparison of the legacy and current view orientations, search for *"View Options: 3D UI Options"* in the HFSS help. Additionally, views associated with **Alt + double-click** zones have been redefined. The current orientations are shown in the help topic, *"Changing the Model View with Alt+Double-Click Areas."*

Verify Solution Type

The parenthetical phrase appended to the name of the design (second entry in the Project Manager) indicates the type of solution that has already been defined for the sample model. If **HFSSDesign*x* (Transient Network)** is shown, then the solution is *Transient* and the *Network Analysis* option is also selected. These settings are the correct ones for this exercise.

Optionally, you can access the *Solution Type* dialog box and verify the settings, as follows:

1. Using the menu bar, click **HFSS** > **Solution Type**.

> **Note:**
>
> Alternatively, under *broadbandhorn* in the Project Manager, right-click **HFSSDesign1 (Transient Network)** and choose **Solution Type** from the shortcut menu.

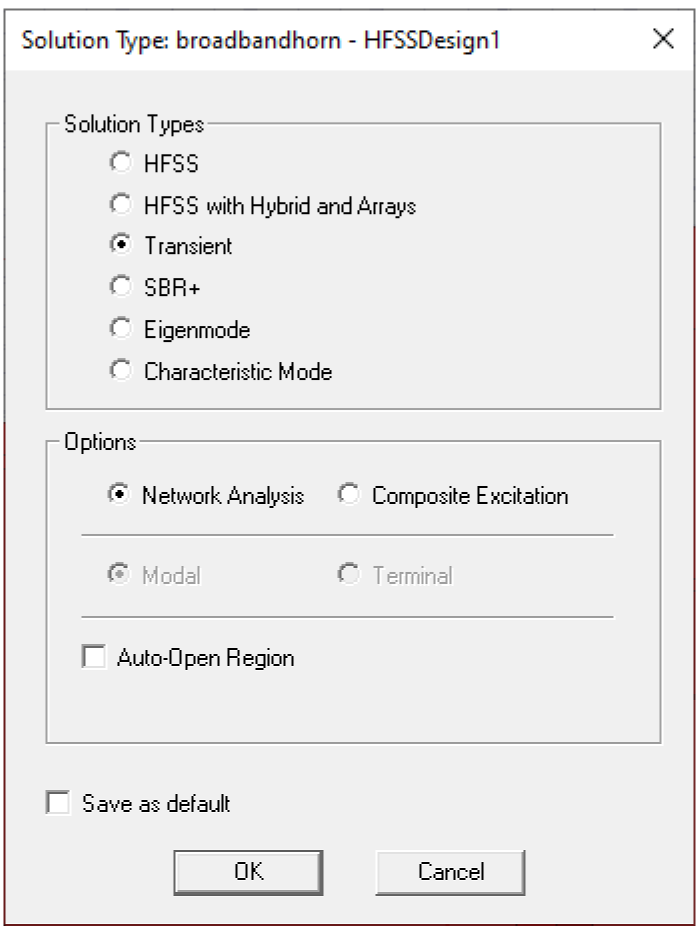


Figure 2-2: Solution Type

2. In the *Solution Type* dialog box that appears, ensure that **Transient** is selected under *Solution Types* and **Network Analysis** under *Options*.
3. Click **OK**.

Assign Material to Horn and Pin

Observe that the **horn** and **pin** objects appear in the History Tree under *Model > Solids > Not Assigned*. Both objects are to be made of copper, and you can define the material for both of them in a single operation, as follows:

1. Under *Model > Solids > Not Assigned* in the History Tree, select **horn** and **pin**.

> **Note:**
>
> After clicking the first item to select it, hold down the **Ctrl** key and click to select additional items.

2. In the docked *Properties* window, click in the *Value* cell of the **Material** row and choose **Edit** from the drop-down menu.
3. In the *Select Definition* dialog box that appears, select **copper** in the materials list.
4. Click **OK**.

   Both objects are now listed under *Model > Solids > copper* in the History Tree.

5. While pressing **Ctrl**, click **pin** in the History Tree to **deselect** it. (The **horn** object must *remain selected*.)
6. In the docked Properties window, change the **Transparent** value to **0.9**.

   During post processing, transparency of the horn antenna will enhance visibility of the E-field overlay you are going to apply to the model.

7. Click in the Modeler window's background area to clear the current selection.

   The model should resemble the figure below:

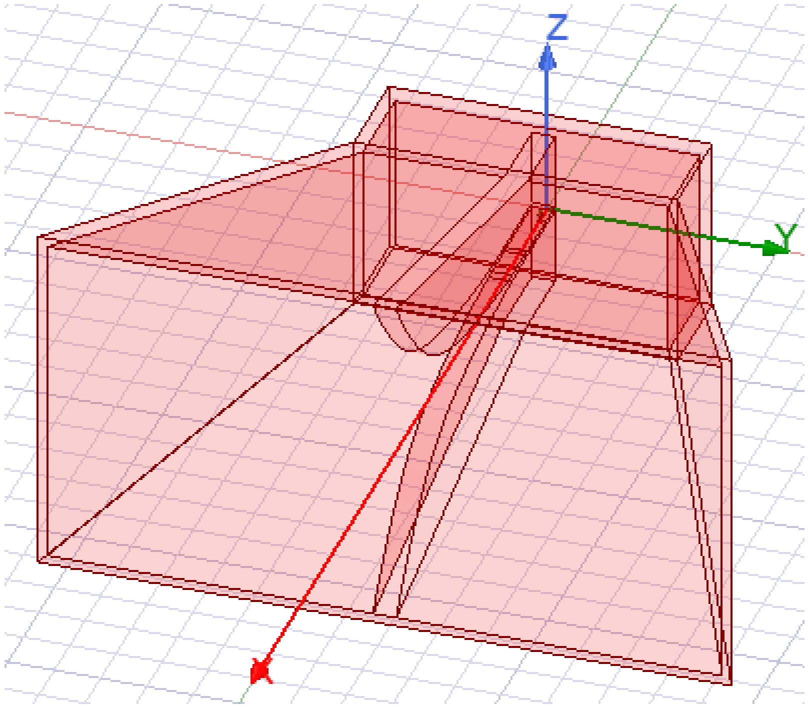


Figure 2-3: Horn Transparency Applied

8. **Save** the project. (This command is available from all ribbon tabs.)

> **Note:**
>
> Even though Ansys Electronics Desktop automatically saves projects at specified intervals (specifically, after every ten edits by default), it's a good idea to save your work frequently.

Create an Open Region

It is not necessary for you to manually create an open region around the model. The process of creating the region and assigning the appropriate boundaries is automated. There are two methods available for creating open regions:

- **Auto-Open Region** option: This option is in the *Solution Type* dialog box. It creates a region of the appropriate size, assigns the specified boundaries (Radiation, FE-BI, or PML), creates default radiation setups, and creates a default analysis setup.
- **Create Open Region** command: This command creates a vacuum region of the appropriate size based on a specified Operating Frequency, assigns the specified boundaries (Radiation or PML), and creates default radiation setups. Optionally, it also applies an

infinite ground plane to the outer face that is in the specified global direction. It does not create a default analysis setup.

You will use the second option for this exercise and specify an *Operating Frequency* of 0.7 GHz and the *PML* (perfectly matched layer) boundary type. The padding (that is, the distance from the model's faces to the open region's boundaries) for the PML boundary option is set as one-fourth of the signal wave length (λ/4).

Create the open region and PML boundaries as follows:

1. Using the menu bar, click **HFSS** > **Model** > **Create Open Region**.
2. In the *Create Open Region* dialog box that appears, specify the settings shown in the following figure and then click **OK**:

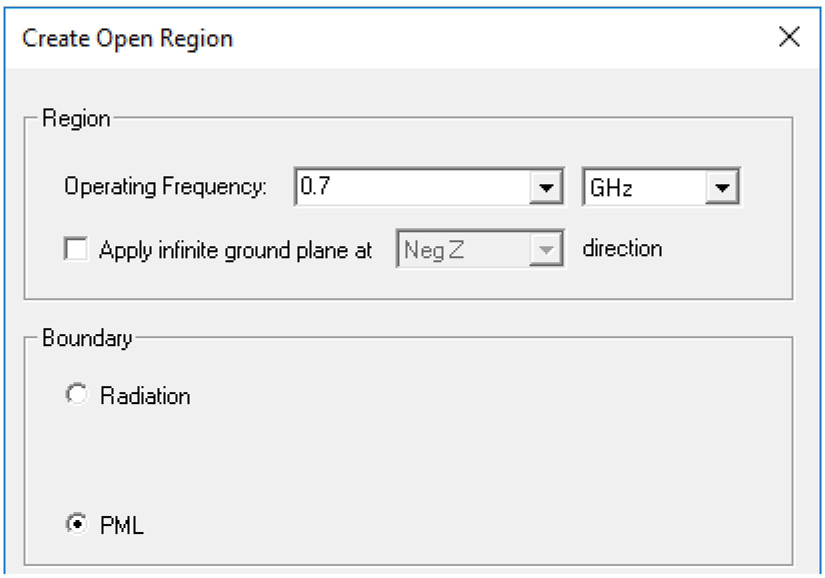


Figure 2-4: Open Region Settings

3. Press **Ctrl+D** to fit the open region to the display area.

   By default, the open region is displayed in red and as a wireframe only (no surface shading):

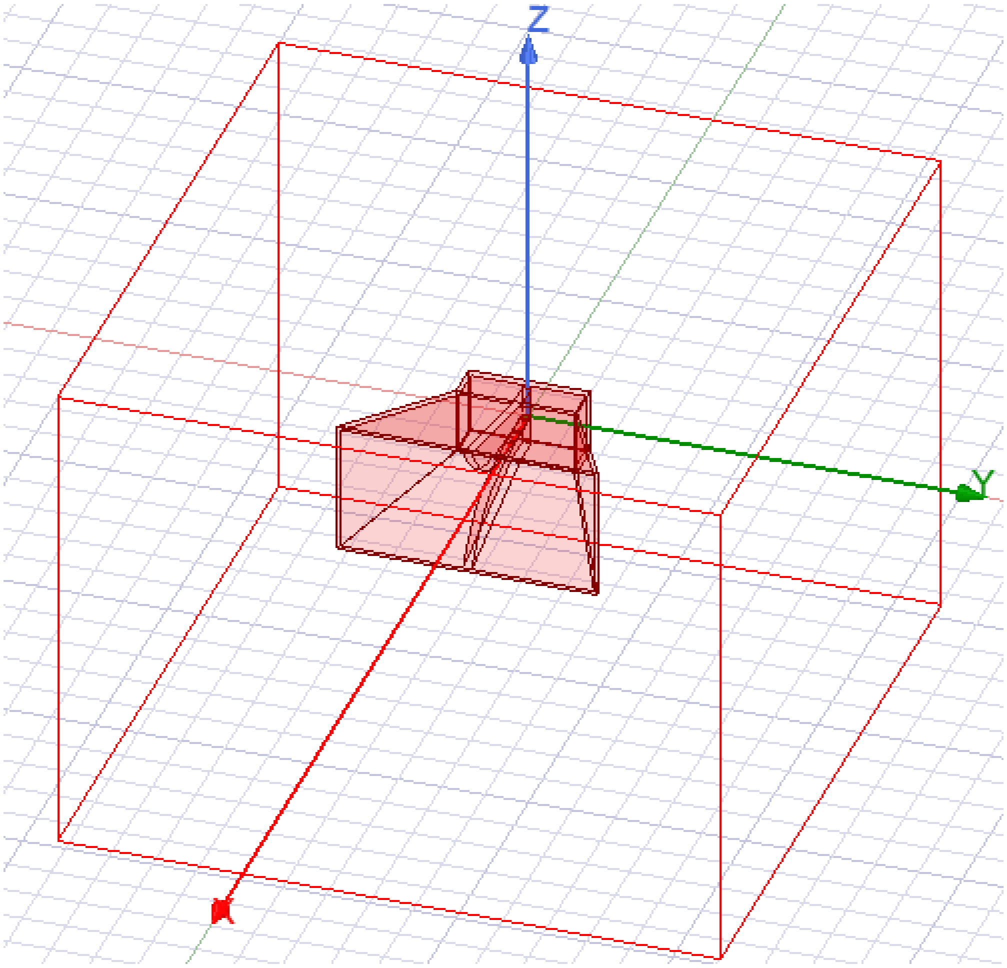


Figure 2-5: Open Region Created

4. Explore the additions to the History Tree under *Model* > *Solids* and *Model* > *PMLGroup1*:
   - Four *PMLGroup1*... sub-branches: If you expand these sub-branches, you will find a total of twenty-six *PML_RadiatingSurface* objects. Select any of them to see the highlighted PML locations around the model.
   - A *vacuum* branch: Expand this branch to see the radiating surface (consisting of the Create Region command and a number of automatically created face coordinate systems and sweep operations). Select the *CreateRegion* entry to see the padding type and offset in all six global directions displayed in the docked *Properties* window.

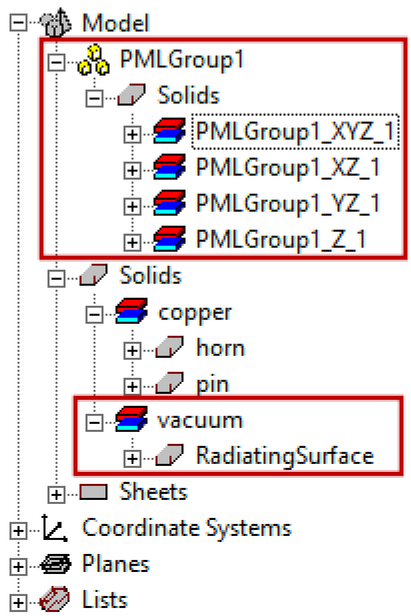


Figure 2-6: Open Region Additions to the History Tree

> **Note:**
>
> For this model, the distance from the model to the radiating surface (that is, the padding) was automatically set to 107.143 mm, which is equal to one-fourth of the wavelength (λ/4) at the specified operating frequency of 0.7 GHz. Additionally, the thickness of the PMLs was automatically set to 217.867 mm. This thickness is determined by the program based on the operating frequency and a geometrical analysis of the model.

5. Expand the **Boundaries** branch in the Project Manager.

   There, you can see a list of the radiation boundaries created as part of the *Create Open Region* operation:

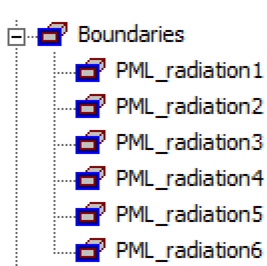


Figure 2-7: Radiation Boundaries

6. Select any of the PML radiation boundaries to see them visualized on the model.

7. Click in the Modeler window's background area to clear any selection.

Create Faces 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. On the **Draw** ribbon tab, ensure that **XY** is selected in the **Drawing plane** drop-down menu.
2. On the **Draw** ribbon tab, click **Draw rectangle**.
3. Press **F4** to use the dialog box entry mode.
4. In the **Command** tab of the *CreateRectangle* dialog box, specify the settings shown in the following figure:

Command

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

Figure 2-8: XY Rectangle Properties – *Command* Tab

5. In the **Attribute** tab of the *CreateRectangle* dialog box, select the **Display Wireframe** option.
6. Click **OK** to create the XY rectangle.
7. On the **Draw** ribbon tab, select **XZ** from the **Drawing plane** drop-down menu.
8. On the **Draw** ribbon tab, click **Draw rectangle**.
9. In the **Command** tab of the *CreateRectangle* dialog box, specify the settings shown in the following figure:

Command

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

Figure 2-9: XZ Rectangle Properties – *Command* Tab

10. In the **Attribute** tab of the *CreateRectangle* dialog box, select the **Display Wireframe** option.
11. Click **OK** to create the XZ rectangle.
12. Clear the current selection.
13. On the **Draw** ribbon tab, click **Grid** (*Toggle grid visibility*) to hide the grid, making the rectangles you just drew more easily seen:

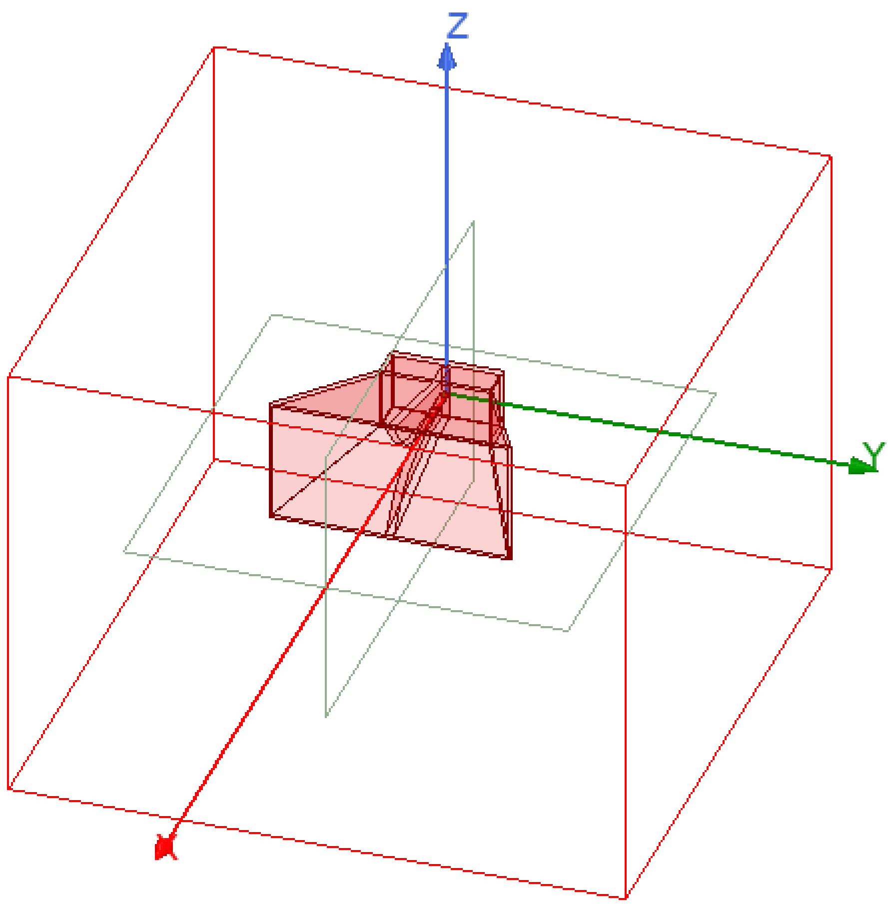


Figure 2-10: Rectangles Added

You need to define a “list” in order to specify later where you wish to save fields:

14. Under *Model* > *Sheets* > *Unassigned* in the History Tree, select **Rectangle1** and **Rectangle2**.
15. From the menu bar, click **Modeler** > **List** > **Create** > **Object List**.

    ObjectList1 appears under *Lists* in the History Tree.

16. Select **ObjectList1** in the History Tree to display its properties in the docked *Properties* window.
17. Change the **Name** to from *ObjectList1* to **PlotFields** and press **Enter**.

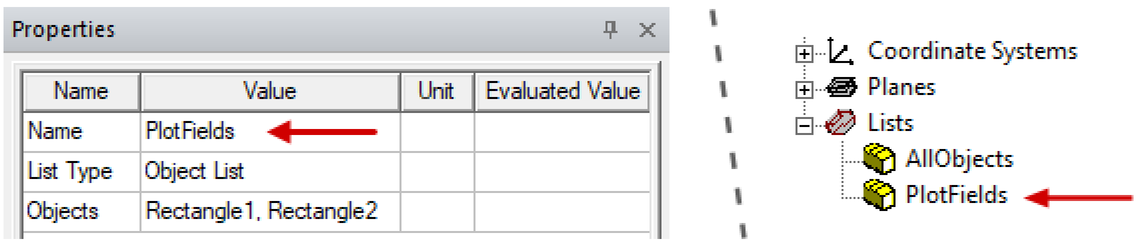


Figure 2-11: Renaming *Objectlist1* as *PlotFields*

18. **Save** the project.

Assign Excitation

You will assign a wave port excitation to the sheet object, *source*. Before doing that, verify that auto-assignment of port terminals is enabled in the program options, as follows:

1. On the **Desktop** ribbon tab, click **General Options**.
2. In the *Options* dialog box that appears, expand the **HFSS** branch and select **Boundary Assignment**.
3. Ensure that the **Auto-assign terminals on ports** option is selected, and click **OK**.

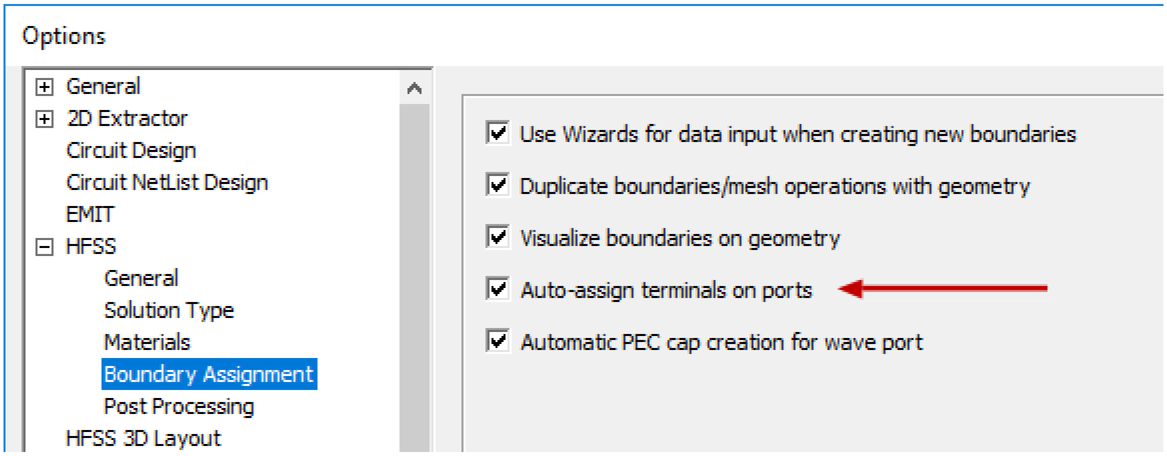


Figure 2-12: Boundary Assignment Options

4. Under *Model* > *Sheets* > *Unassigned* in the History Tree, select **source**.
5. On the **Draw** ribbon tab, click **Fit Selected**. Then use the mouse wheel to zoom out somewhat to get a better view of the context.

> **Tip:**
>
> If you don't have a three-button mouse with a zoom/scroll wheel, use a ribbon command: On the **Draw** ribbon tab, click **Zoom** > **Zoom Out** once or twice.

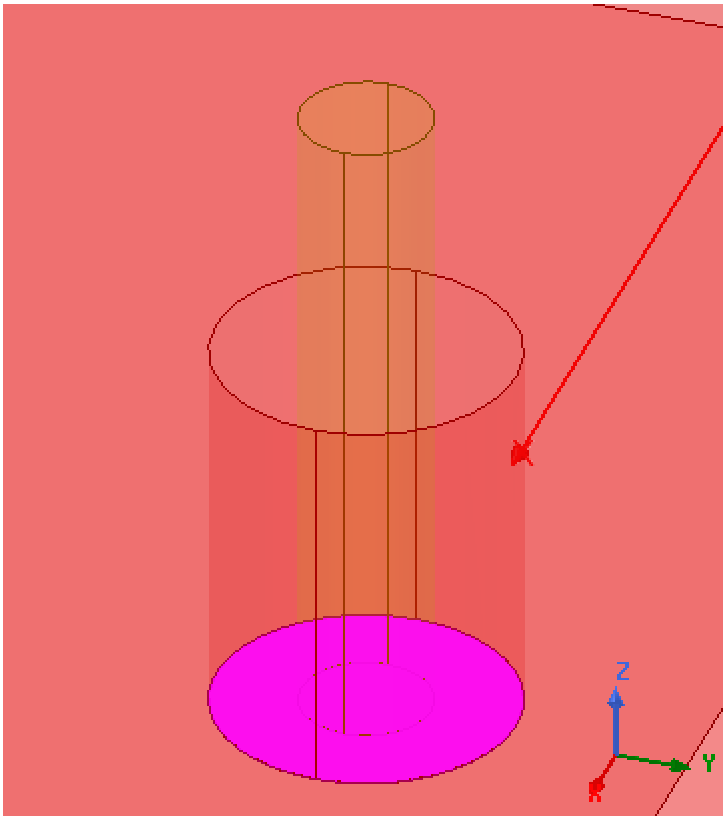


Figure 2-13: Source Location

> **Note:**
>
> This object serves as the source of a coaxial cable in which:
>
> - The outer conductor connects to one antenna ridge (in this case, the bottom one).
> - The inner conductor connects to the opposite ridge (in this case, the top one).

6. Right-click in the Modeler window and select **Assign Excitation** > **Port** > **Terminal Wave Port**.
7. In the *Reference Conductors for Terminals* dialog box that appears, select the **Use as Reference** option in the **horn** row. Therefore, the *pin* will be the terminal:

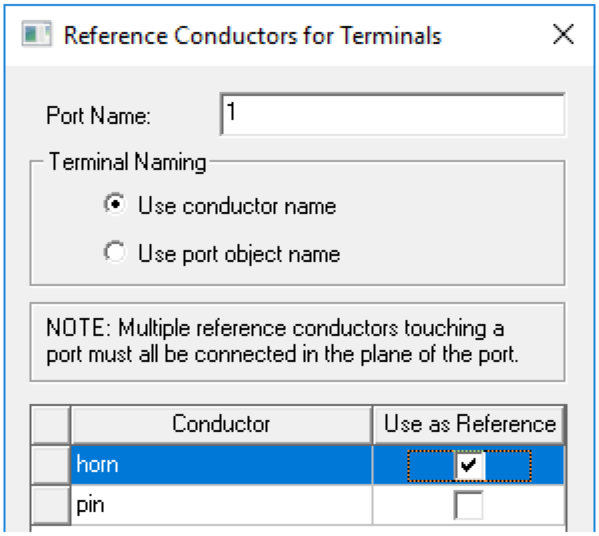


Figure 2-14: Specify the Reference Conductor

8. Click **OK**.
9. Under *Excitations* > *1* in the Project Manager, select **pin_T1**.
10. Rotate the model viewpoint for an underside view to see the terminal associated with the wave port:

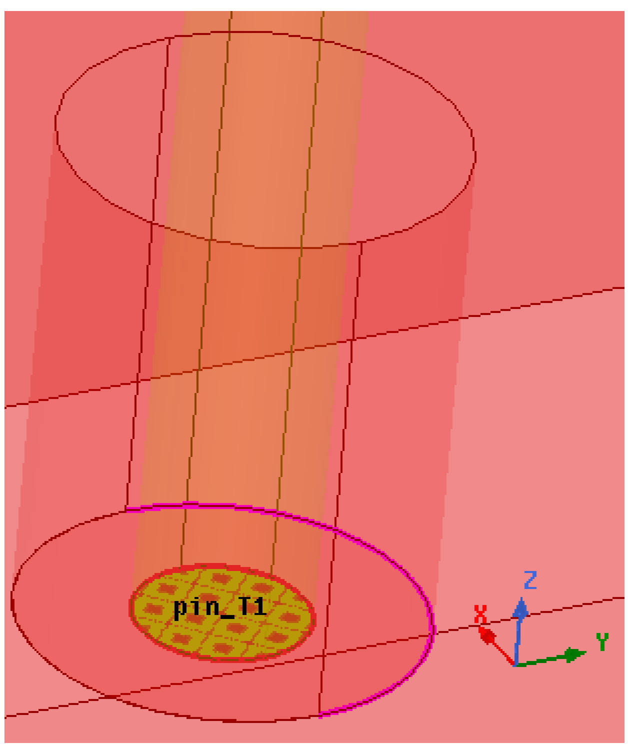


Figure 2-15: Wave Port Terminal

11. On the **Draw** ribbon tab, click **Orient** to restore the default *Trimetric* view of the model. (You don't have to access the *Orient* drop-down menu because *Trimetric* is the default orientation option.)
12. Press **Ctrl+D** to fit the model view.
13. Clear the selection.
14. **Save** the project.

Explanation of Active Ports

In order to obtain the full S-matrix in a *Transient Network Analysis* design with multiple ports, the ports are ON one-at-a-time for a full 3D simulation. Each simulation with a particular port ON produces one column of the S matrix. When you specify a port as passive, it only acts as a termination, saving simulation time but at the expense of not generating the particular column of S-parameters.

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

In either cases, the distinction between active and passive ports provides flexibility.

3 - Simulation and Results

This chapter describes how to set up the transient simulation for the horn antenna and evaluate the results.

Solution Setup

1. On the **Simulation** ribbon tab, click  **Setup** (*Add Solution Setup*).

   The *Transient Solution Setup* dialog box appears, and the *General* tab is initially displayed.

   The mesh for the transient simulation is generated by a regular frequency-domain simulation. For that simulation, the software will determines the appropriate frequency at which to perform the adaptive passes. It uses mixed element orders and the iterative solver. Accept the defaults under the **General** tab, but keep the dialog box open:

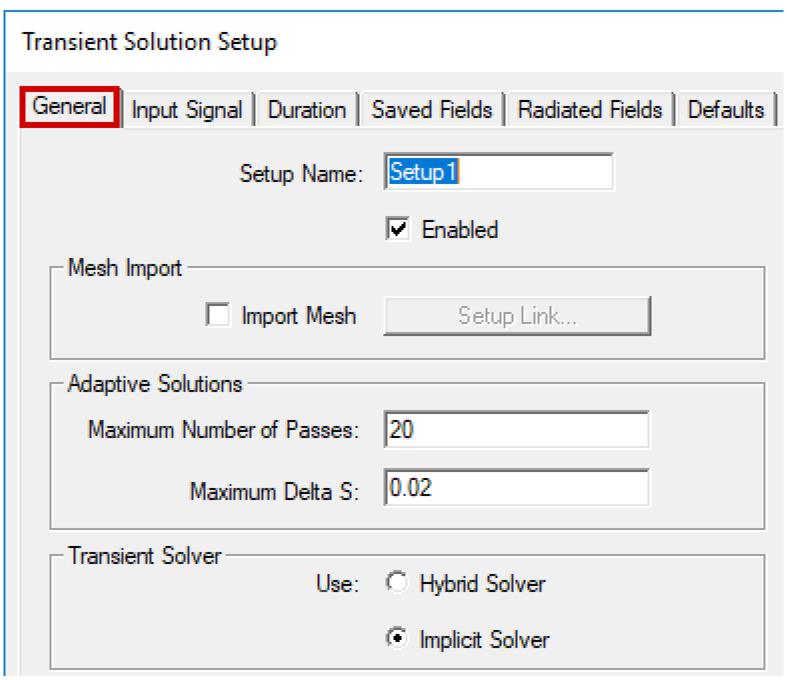


Figure 3-1: Transient Solution Setup – *General* Tab

2. Under the **Input Signal** tab, specify the frequency band of interest, as shown below:

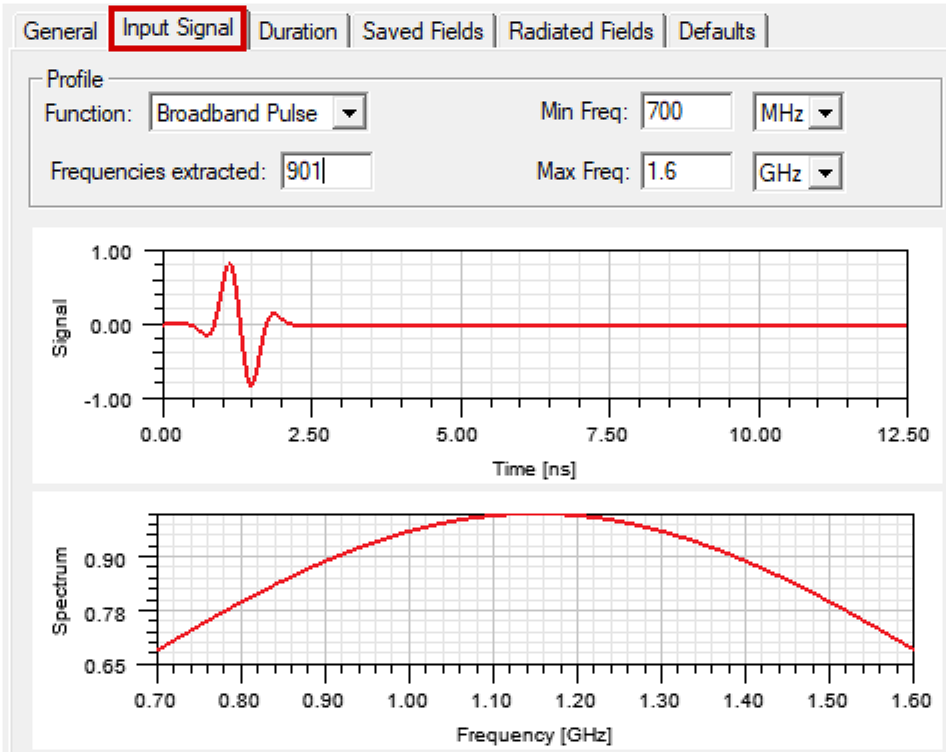


Figure 3-2: Transient Solution Setup – *Input Signal* Tab

> **Note:**
>
> The specified frequency band is covered by a modulated Gaussian pulse in the time domain, as can be seen in the upper graph on the *Input Signal* tab.

3. Under the **Duration** tab, specify limits for the simulated time of the transient analysis, as shown below:

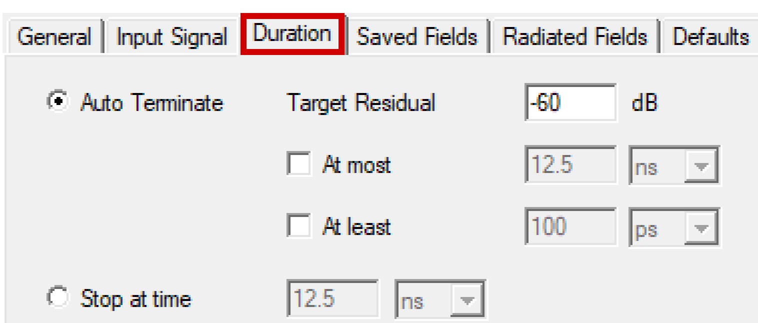


Figure 3-3: Transient Solution Setup – *Duration* Tab

> **Note:**
>
> 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 peak field has fallen to one-thousandth (0.001) of its all-time high (or -60 dB), the simulation is stopped. Optionally, you can specify *At most* and/or *At least* duration limits. The default *Auto Terminate* behavior is appropriate for most transient simulations. Only in special cases would you find it necessary to impose an *At most* or *At least* duration limit.

4. Under the **Saved Fields** tab, select the object list that you defined previously (**PlotFields**) and specify that you want to **Save fields every 30 ps**:

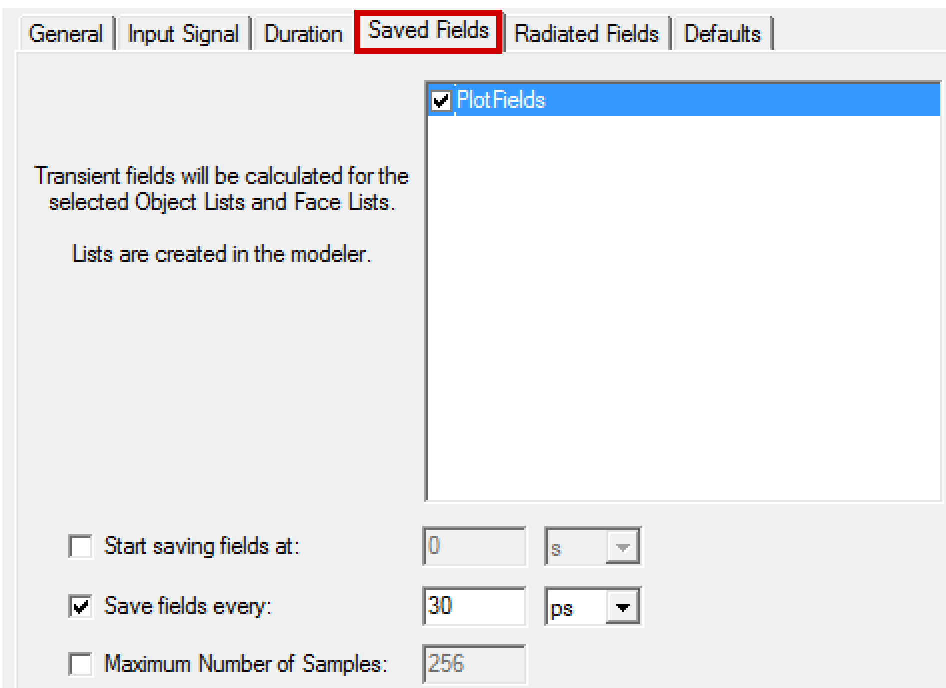


Figure 3-4: Transient Solution Setup – *Saved Fields* Tab

> **Note:**
>
> The 30 picoseconds field saving interval is a small-enough fraction of the broadband pulse duration to provide a smooth field animation later on.

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

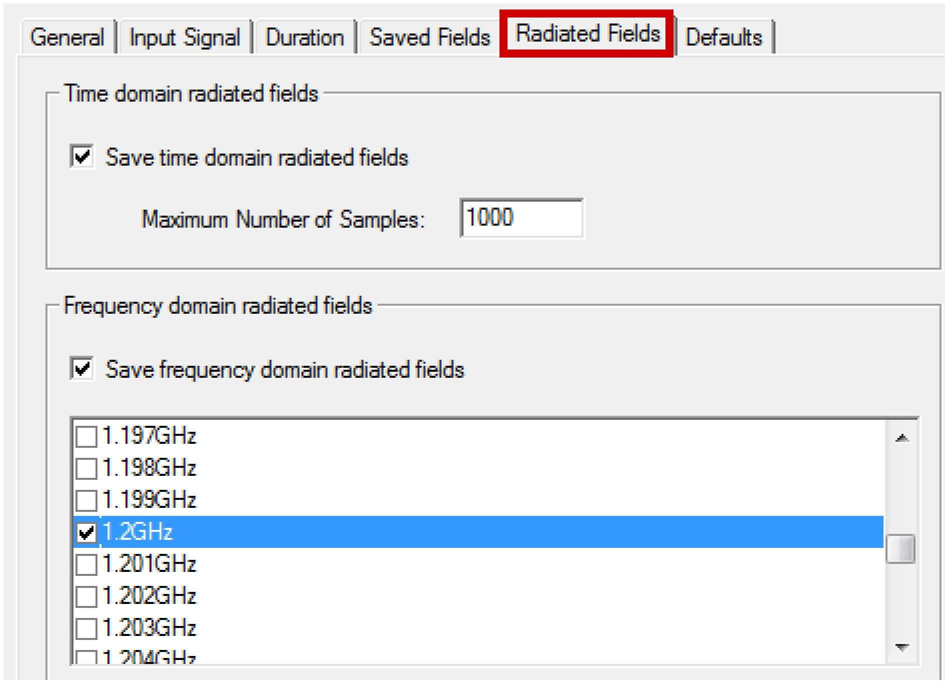


Figure 3-5: Transient Solution Setup – *Radiated Fields* Tab

6. The *Transient Solution Setup* is complete. Click **OK**.

Validate and Run Simulation

1. On the **Simulation** ribbon tab, click **Validate**.

   The *Validation Check* window appears and, if you've set up the model correctly, it should show no warnings or errors:

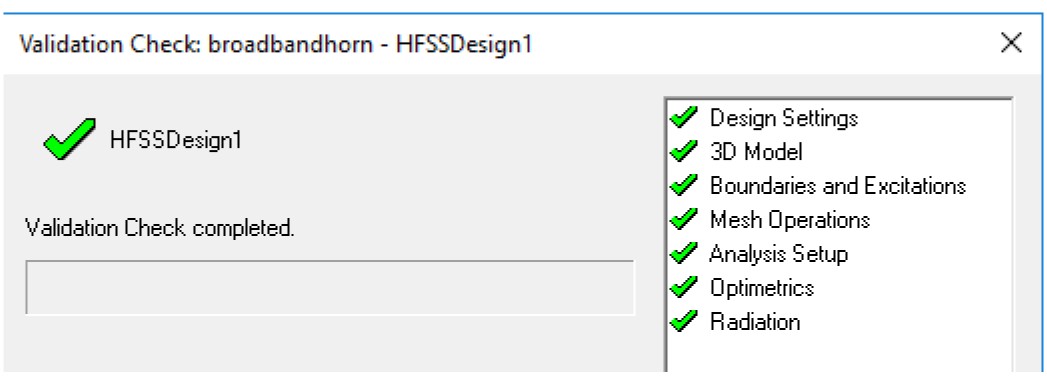


Figure 3-6: Validation Check

2. Click **Close**.

3. On the **Simulation** ribbon tab, click **Analyze All**.

   The progress bar is based on the maximum simulated time from the *Duration* tab in the *Transient Solution Setup* dialog box. If no duration is specified, progress is based on 20× (model diagonal)/(speed of light). In general, the simulation is done well before the progress bar has reached its maximum length.

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

   - Observe the convergence data. (On the **Results** ribbon tab, click **Solution Data** and choose the **Convergence** tab of the *Solutions* dialog box.)
   - Create a transient plot (for example response or residual vs. time), and watch the plot develop as the solution progresses.

   In the next topic you will generate two rectangular plots. This simulation takes under three minutes to solve on a reasonably current computer workstation. The full curve will not appear on the first plot until the solution has finished.

Transient Plots

Plot the *Input* and *Output* traces in the first report and plot *Residual* in dB20 in the second. You will also modify the Y axis units for the *Input* and *Output* plot.

1. On the **Results** ribbon tab, click **Terminal Solution Data Report** > **2D**.

   The *Report* dialog box appears.

2. Specify the settings shown in the following figure:

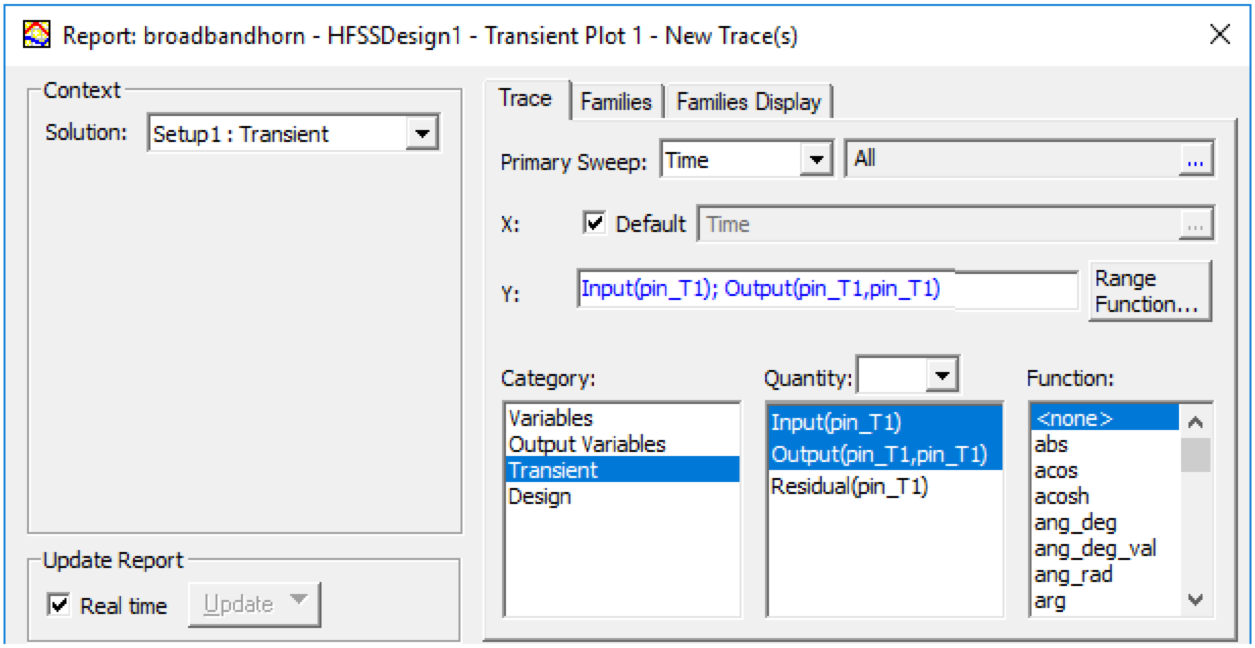


Figure 3-7: Input and Output Transient Plot Settings

3. Click **New Report**, but do not close the dialog box yet.

   Transient Plot 1 appears in a new window. (Move the *Report* dialog box to the side or move the plot window for an unobstructed view.)

4. Double-click within the **Transient Plot 1** window to access the plot properties.
5. In the *Properties* dialog box that appears, select the **Y1 Scaling** tab and make the following changes:
   a. Under *Manual Units*, clear the **Auto Units** option.
   b. Click in the *Value* column of the **Units** row and choose **V** from the list.

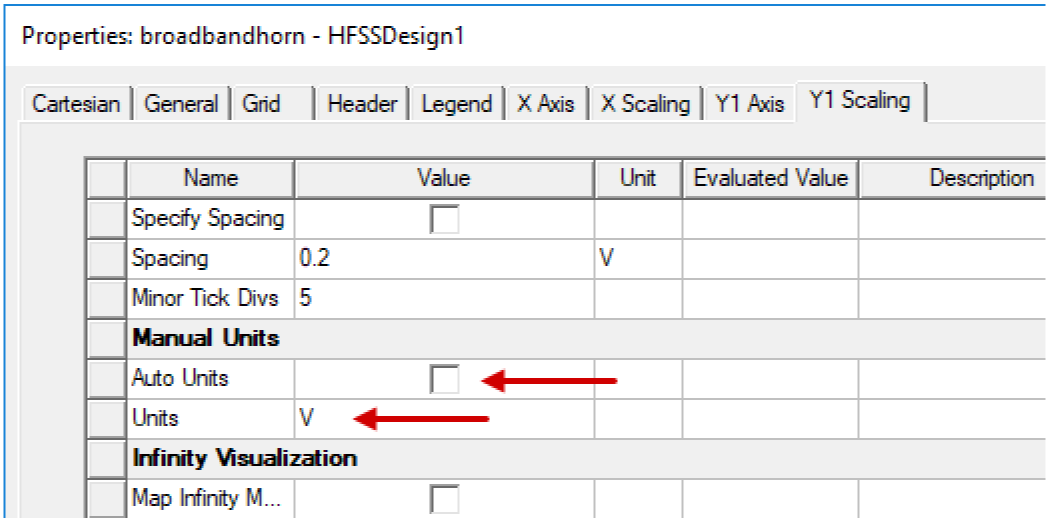


Figure 3-8: Modifying the Plot's Y-Axis Units

6. Click **OK**.

   The modified plot should look like the following figure:

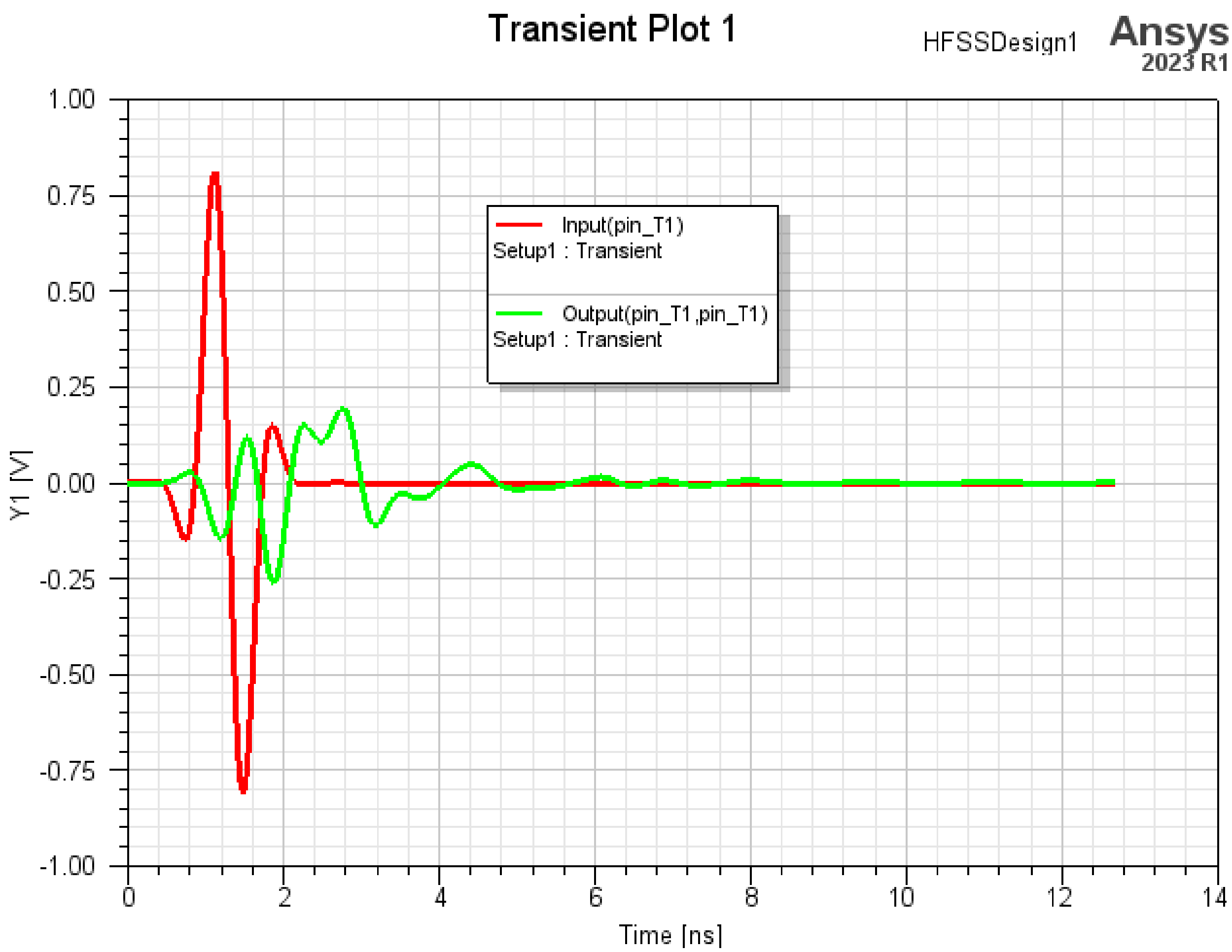


Figure 3-9: Input and Output Transient Plot

> **Note:**
>
> The *Input* and *Output* traces respectively show, as a function of time, the excitation at the port and the reflection of the excitation at the same port. Notice that the reflection is inverted and delayed about 0.1 ns relative to the excitation. The shape of the reflection trace is more complex than that of the excitation trace due to the interaction of the electromagnetic wave and the geometry of the antenna.

7. In the *Report* dialog box, specify the settings shown below:

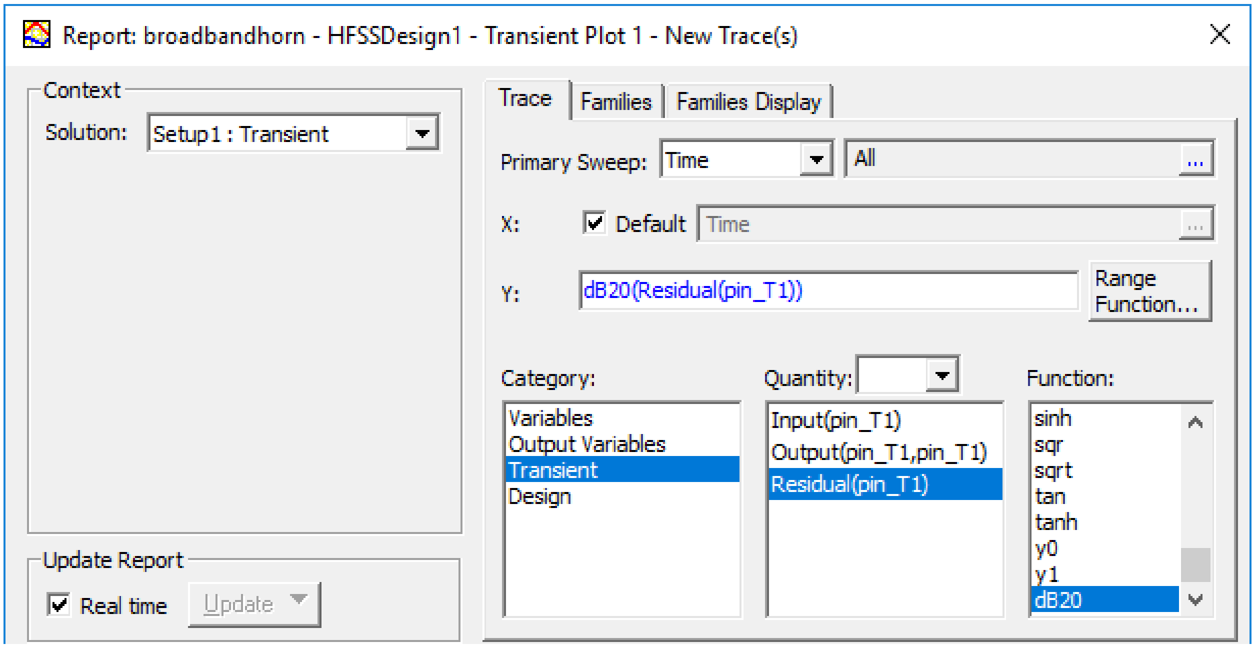


Figure 3-10: Residual Plot Settings

8. Click **New Report** but keep the dialog box open.

   Transient Plot 2 appears in a new window:

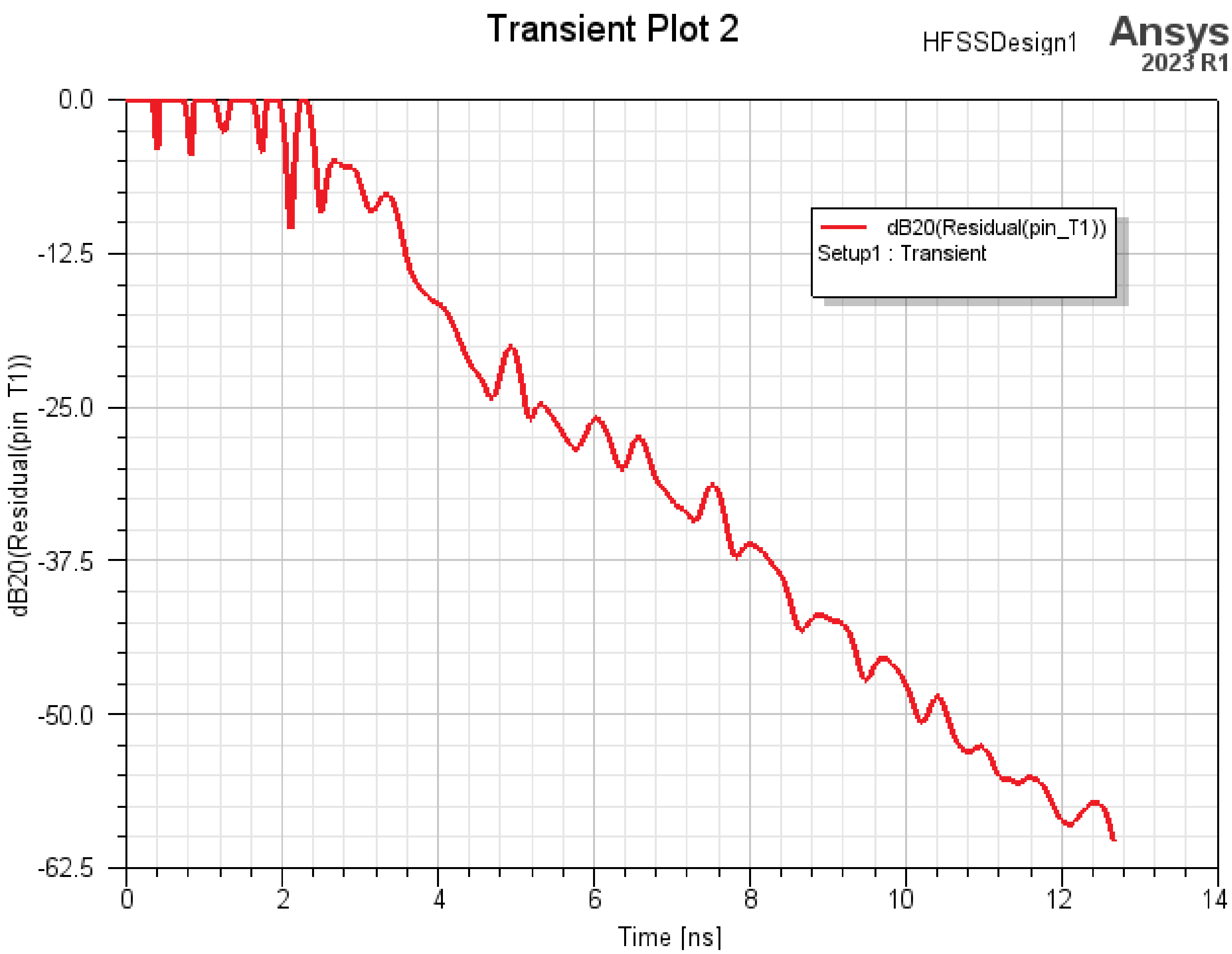


Figure 3-11: ***Residual Transient Plot***

> **Note:**
>
> The *Residual* trace shows the maximum field level in the model as a function of time. The simulation is complete when the residual field falls below 0.001 (-60 dB) of its all-time high.

S-Parameter vs. Frequency

Your next plot will be in the frequency domain rather than the time domain. Since this model only has a single port, there is only one S-Parameter available to plot, S_{11}.

The report dialog box should still be open from the transient plot creation procedure. If not, repeat step 1 from the previous topic before proceeding to the steps below.

1. In the *Context* section of the *Report* dialog box, choose **Setup1 : Spectral** from the **Solution** drop-down menu:

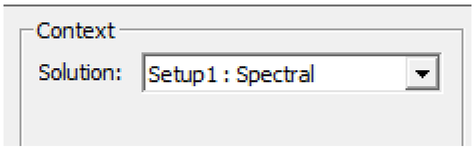


Figure 3-12: *Solution* Option for a Frequency Domain Plot

2. In The *Trace* tab of the *Report* dialog box, specify the settings shown in the following figure to plot the magnitude of S_{11}:

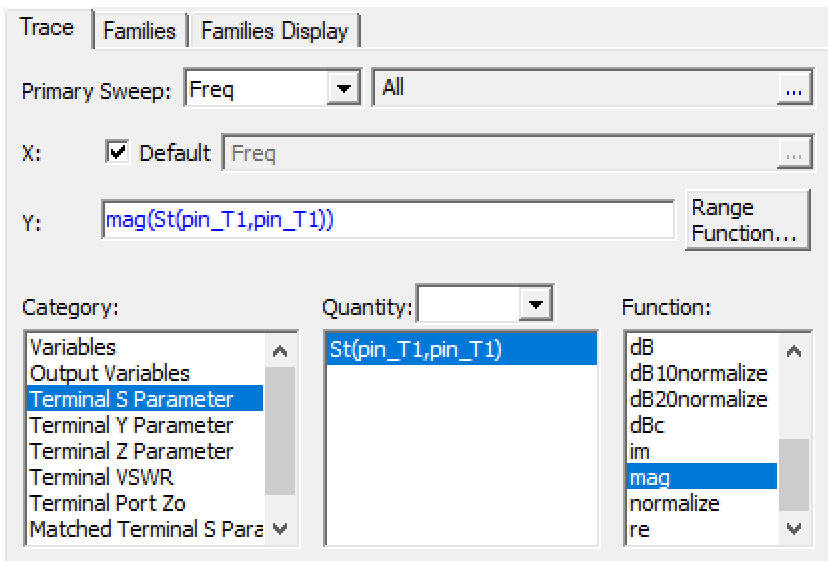


Figure 3-13: S=Parameter vs. Frequency Plot Settings

3. Click **New Report** and then **Close**.

   Terminal S Parameter Plot 1 appears in a new window:

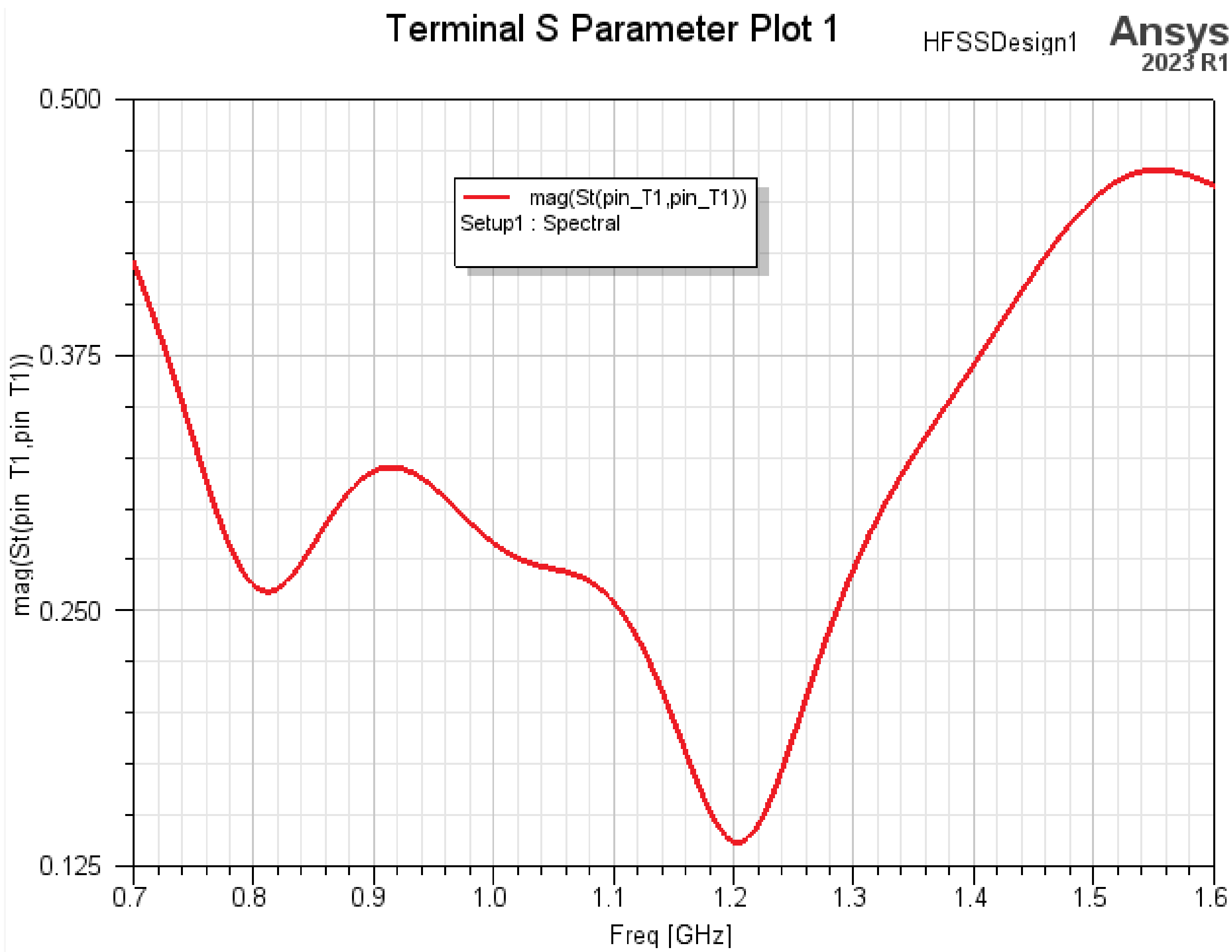


Figure 3-14: S-Parameter vs. Frequency Plot

> **Note:**
>
> This plot shows the signal reflection magnitude at the antenna port as a function of frequency. The minimal reflection occurs at a frequency of about 1.2 GHz.

Field Overlay

Next, you will plot the E-field magnitude at a specified time during the transient analysis. You will overlay the fields on the previously specified pair of rectangles in the *PlotFields* list. To provide an optimal view of the fieldsl, you will hide the open region and change the model viewpoint.

1. Under *Lists* in the History Tree, select **PlotFields**.

   The two rectangular objects comprising the list are highlighted.

2. In the Project Tree, right-click **Field Overlays** and choose **Plot Fields** > **E_t** > **Mag_E_t** frm the context menu.
3. In the *Intrinsic Variables* section of the *Create Field Plot* dialog box that appears, select an arbitrary **Time** somewhat after the input pulse has been completed (to allow a little field propagation time). Let's try 2.58 ns, which is approximately 500 ps beyond the time at the completion of the pulse.

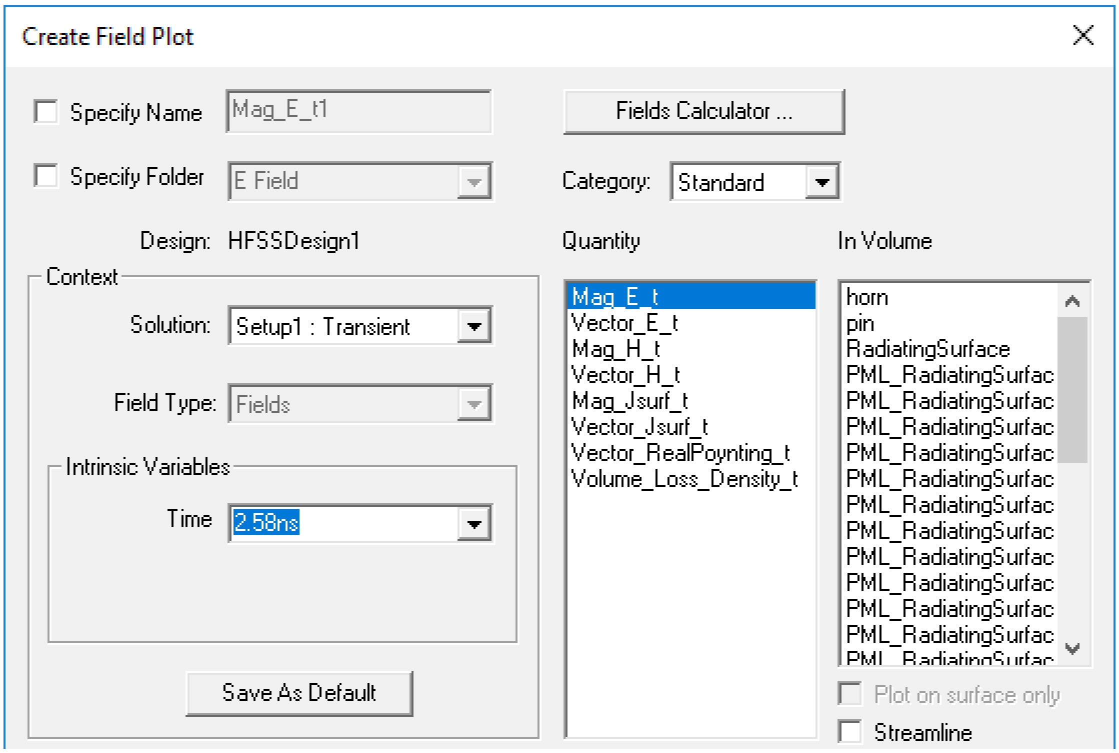


Figure 3-15: Field Overlay Settings

> **Note:**
>
> You can adjust the time later by right-clicking the overlay name (*Mag_E_t1*) in the Project Manager and selecting **Modify**.

4. Click **Done**.

   The field overlay and legend appear in the Modeler window.

5. Under *Model* > *Solids* > *vacuum* in the History Tree, right-click **RadiatingSurface** and choose **View** > **Hide in Active View**.
6. On the **Draw** ribbon tab, choose **Dimetric** from the **Orient** drop-down menu.

7. Double-click within the perimeter of the E Field legend to display the plot properties.
8. On the **Scale** tab, select the **Use Limits** option and specify the following values, keeping the *Units* as *V_per_meter*:
    a. **Min** = **0**
    b. **Max** = **12**

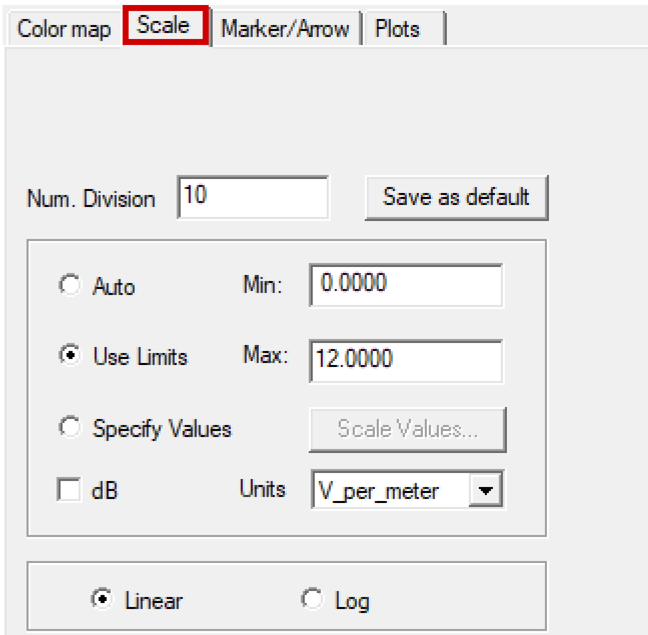


Figure 3-16: Overlay Plot Settings – *Scale* Tab

9. On the **Plots** tab, make the following changes:
    a. Choose **Gourard** from the **IsoValType** drop-down menu.
    b. Choose **Very Fine** from the **Plot quality** drop-down menu.

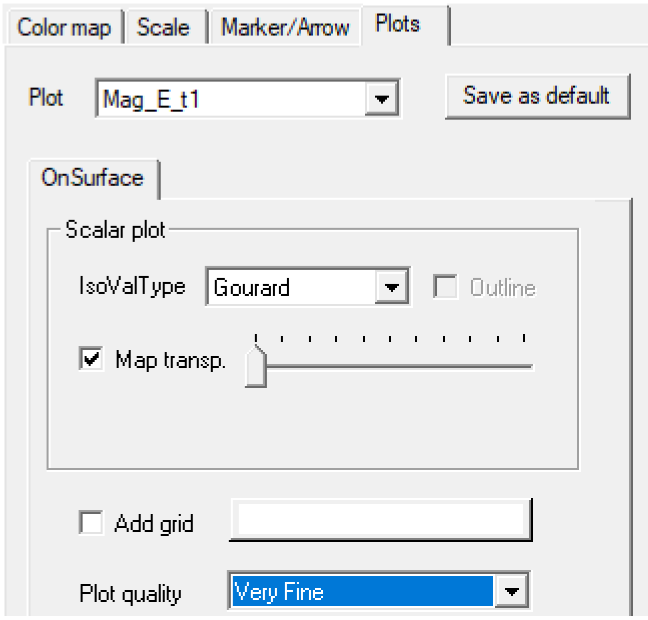


Figure 3-17: Overlay Plot Settings – *Plots* Tab

10. Click **Close**.

    The modified E-Field overlay should now look like the following figure:

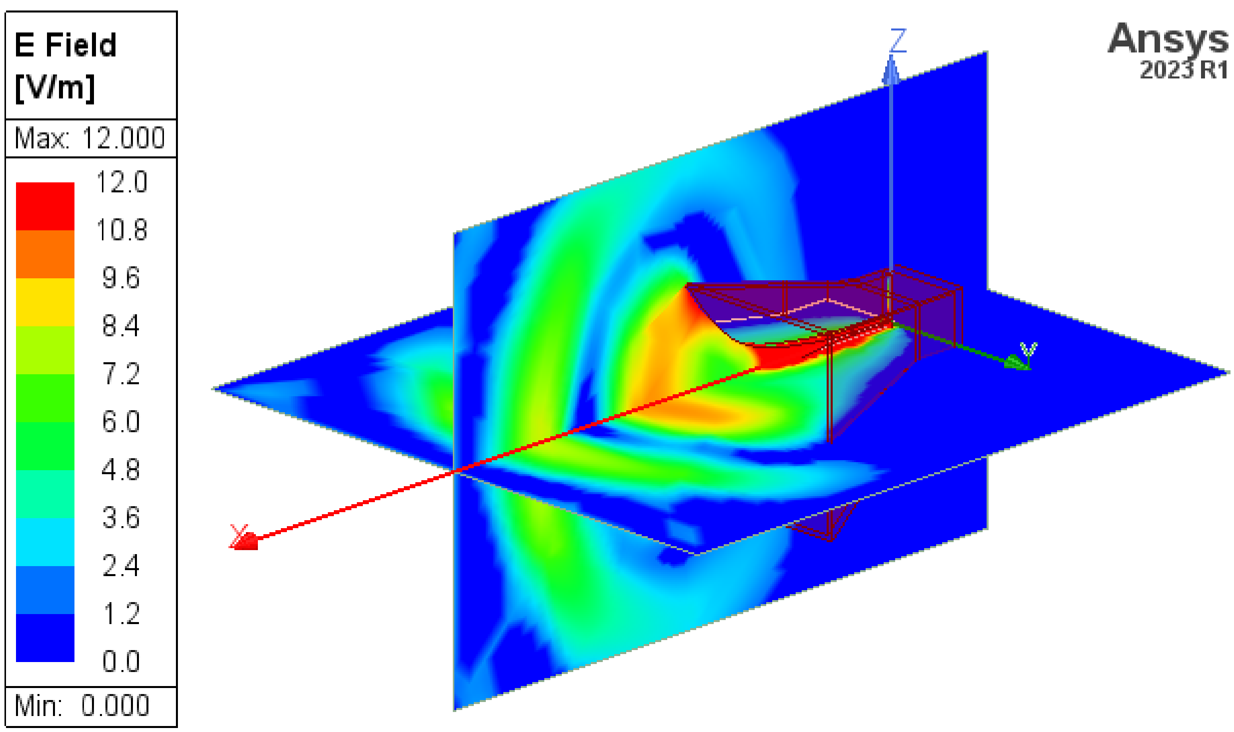


Figure 3-18: E-Field Overlay at Time = 2580 ps

11. Under *Field Overlays > E Field* in the Project Manager, right-click **Mag_E_t1** and select **Animate**.
12. In the *Create Animation Setup* dialog box that appears, specify the settings shown below:

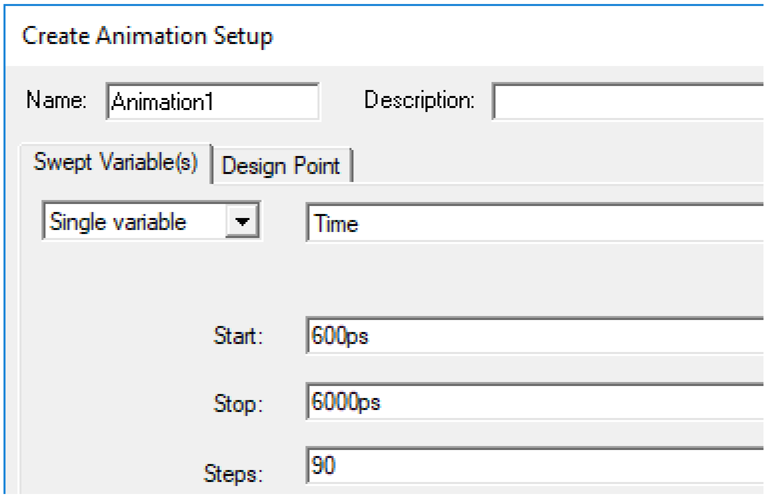


Figure 3-19: ***Setup Animation window***

> **Note:**
>
> The specified 5400 ps time period encompasses the field development, propagation, and decay while excluding portions of the solution that are not very interesting. The transient solution increment was 30 ps, so there are 180 calculated fields in this range (5400 ps / 30 ps). At a setting of 90 Steps, every other calculated field in the range becomes an animation frame, reducing the number of steps to control the animation size and calculation time. However, if necessary, HFSS will interpolate between saved field solutions. So you don't have to specify *Start* and *Stop* times that correspond exactly with calculation points. You also don't have to specify the number of *Steps* such that the animation frames are produced using time increments that are exact multiples of the calculation increment (in this case, 30 ps).

13. Click **OK**.

After a delay while animation frames are being computed, the *Animation* dialog box appears, and the field animation begins to play:

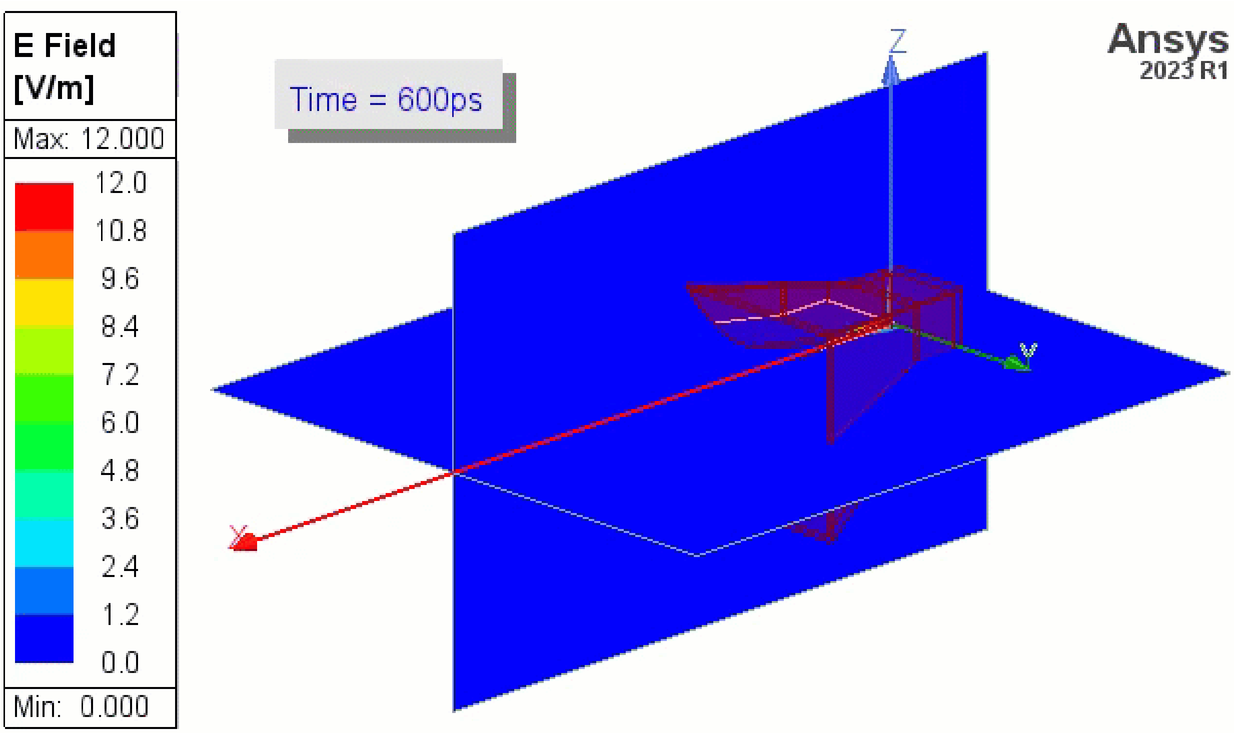


Figure 3-20: E-Field Animation

14. Use the available animation controls to stop, resume, reverse, or control the playback speed of the animation. You can also drag the slider while the animation is stopped to examine any individual frame.
15. Click **Close** when you've finished observing the animation.

Radiated Fields

As part of the open region creation process, the program automatically defined three default *Radiation* setups (*3D, Azimuth,* and *Elevation*). However, none of these options are suitable for this exercise. Therefore, you will modify one of the automatic setups and delete the other two. The *Phi* angle will be fixed at 0°, and *Theta* will vary from 0°–180° in 5° steps.

Revise Far Field Setups:

1. Under *Radiation* in the Project Manager, do the following:
   a. Select **Azimuth** and press **Delete**.
   b. Select **Elevation** and press **Delete**.
   c. Right-click **3D** and choose **Properties** from the shortcut menu.

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

2. Specify the setup *Name* and the range of interest, as shown in the following figure. Then click **OK**:

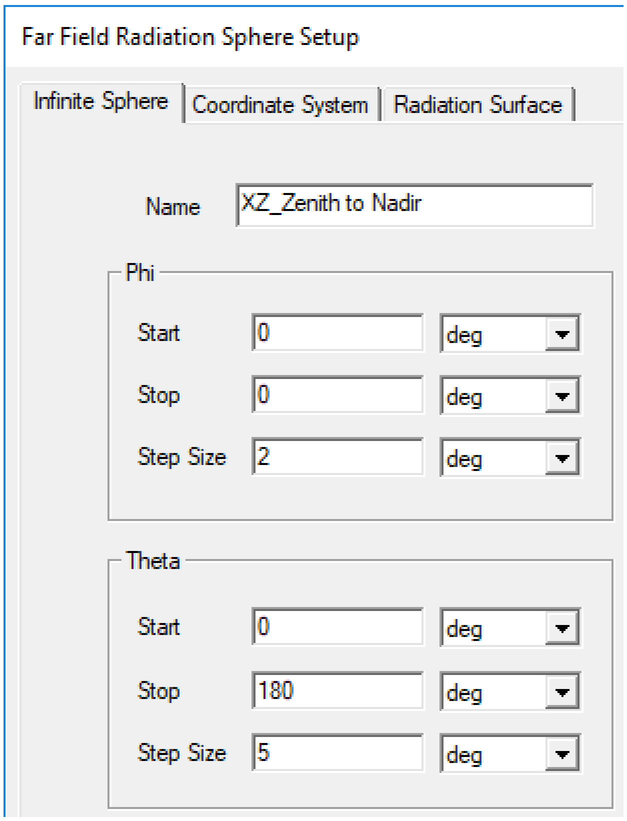


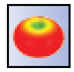
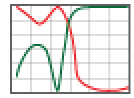
Figure 3-21: Modifying the Radiation Sphere Setup

> **Note:**
>
> This range is in the XZ plane, and spans from zenith to nadir in the *forward* direction of the antenna.

3.  **Save** the project.

Create Radiation Plots:

Finally, you will create two radiation plots, one in the time domain and one in the frequency domain, as follows:

4. On the **Results** ribbon tab, click  **Far Fields Report** >  **2D**.
5. In the *Report* dialog box that appears, specify the settings shown below:

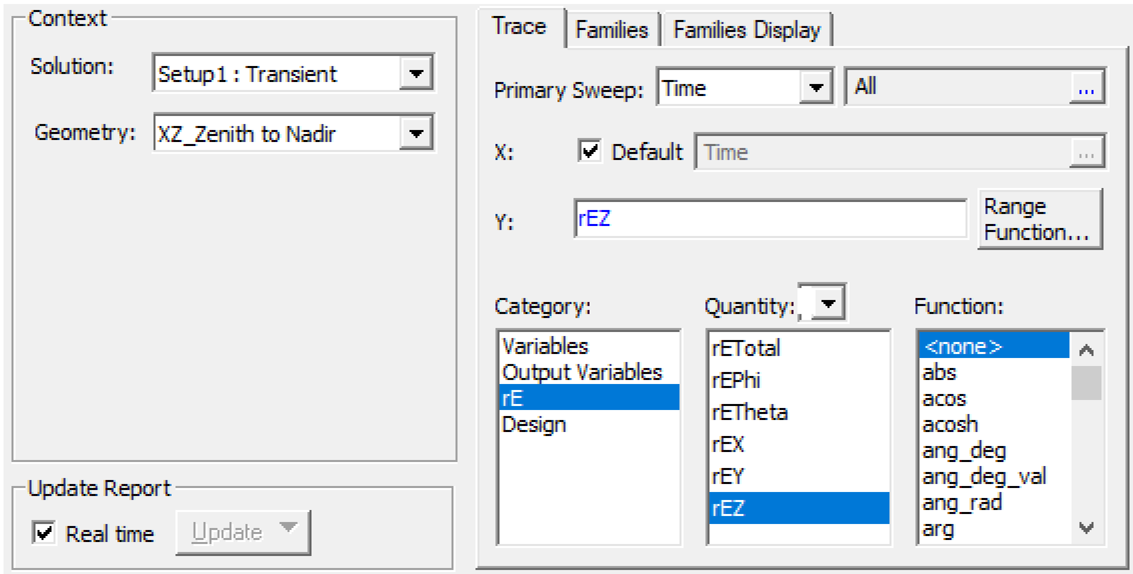


Figure 3-22: Radiation Plot Settings – *Trace* Tab

6. Under the **Families** tab, specify **Theta** = **90deg** and **Phi** = **0deg** (the center of the main beam).

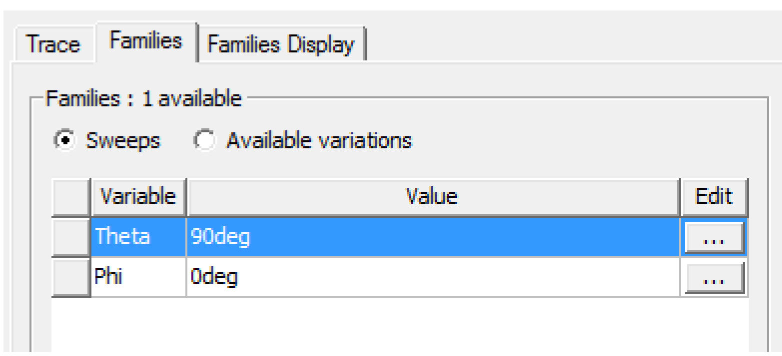


Figure 3-23: Radiation Plot Settings – *Families* Tab

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 multiplied, 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.

7. Click **New Report** but leave the dialog box open.

rE Plot 1 appears in a new window:

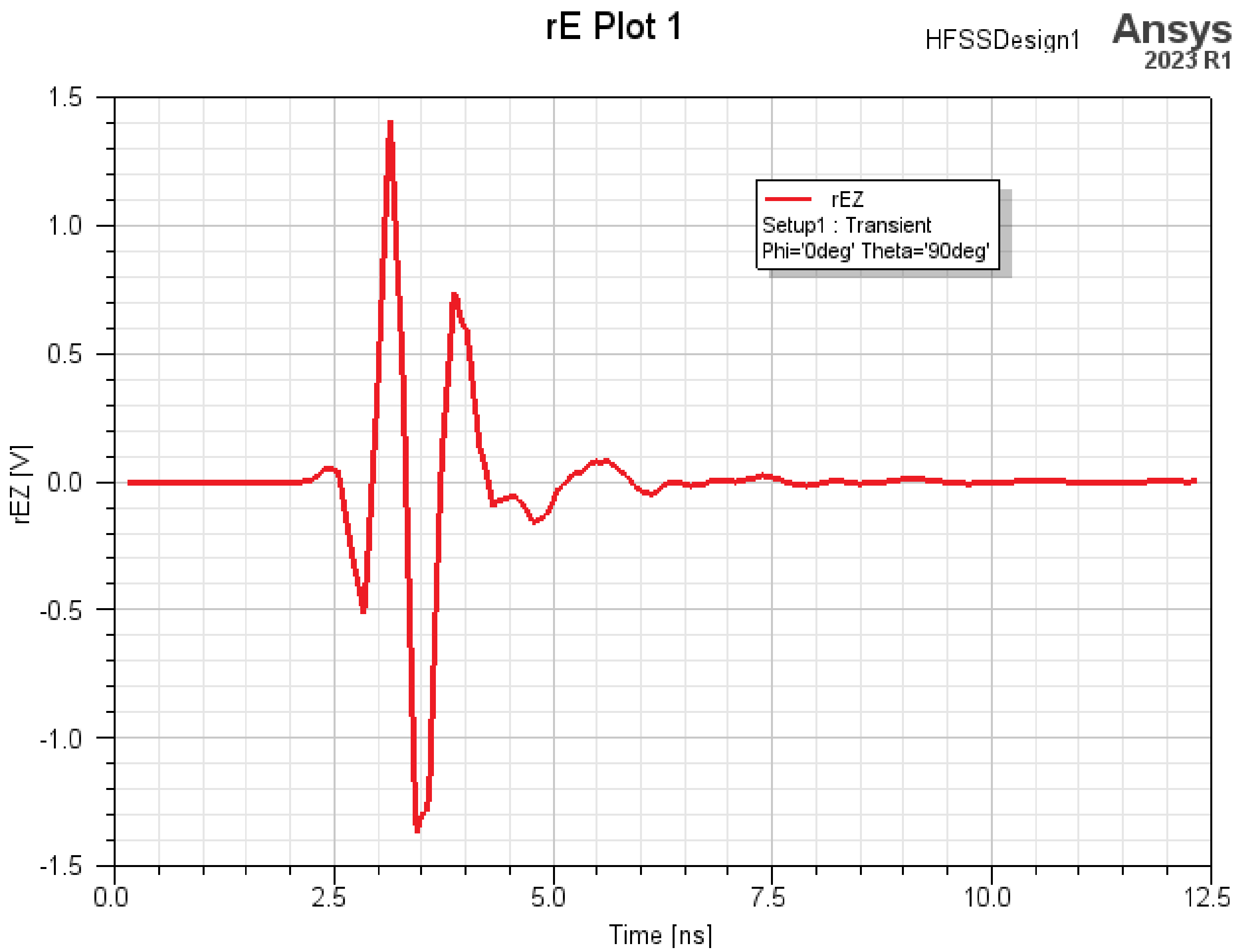


Figure 3-24: Radiated Field Plot

Note:

The preceding figure shows the variation, in the time domain, of the far radiated field along the main beam direction, which can be compared to the pulse applied to the antenna port. A plot like this is instrumental in determining how the original excitation is distorted by the antenna. This information can be used to improve the antenna design (for example, 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.

Next, you will create a frequency domain plot of the radiated field, which will show the variation in the rE_Z field as Theta varies from 0° to 180°.

8. In the *Report* dialog box, make the following changes:
    a. Choose **Setup1 : Spectral** from the **Solution** drop-down menu.
    b. On the **Trace** tab, choose **Theta** from the **Primary Sweep** drop-down menu.
    c. Specify the **Category, Quantity,** and **Function** options as shown below:

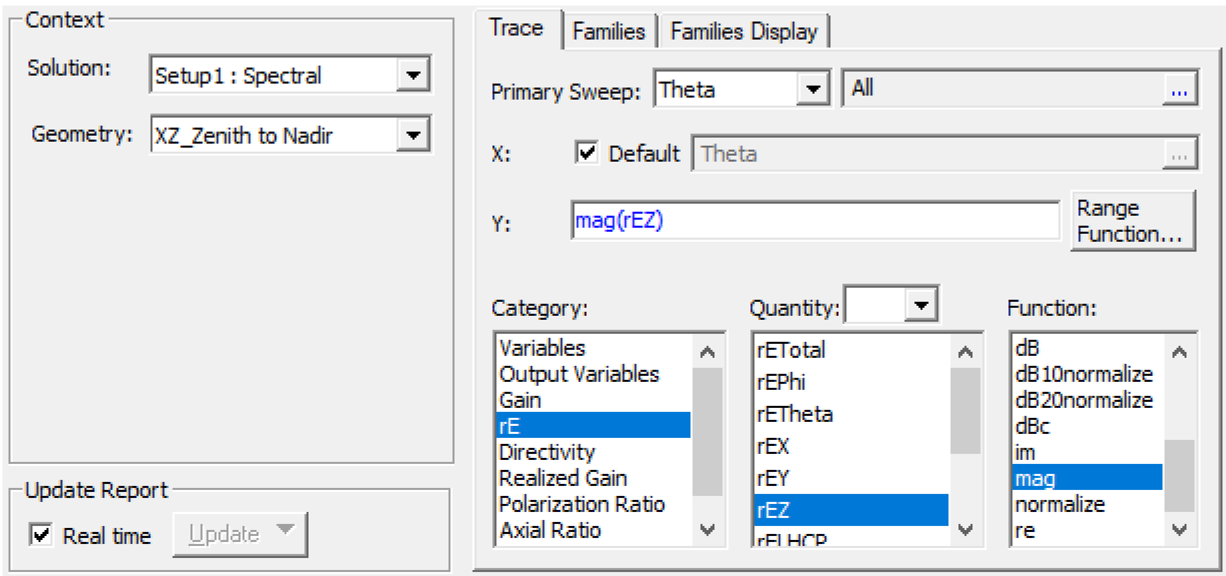


Figure 3-25: Spectral Radiation Plot Settings – *Trace* Tab

> **Note:**
>
> Under the **Families** tab, you will notice that the frequency of *1.2 GHz*, specified earlier under the *Radiated-Fields* tab of the *Transient Solution Setup*dialog box, is automatically specified. This is the only available frequency at which fields were saved.

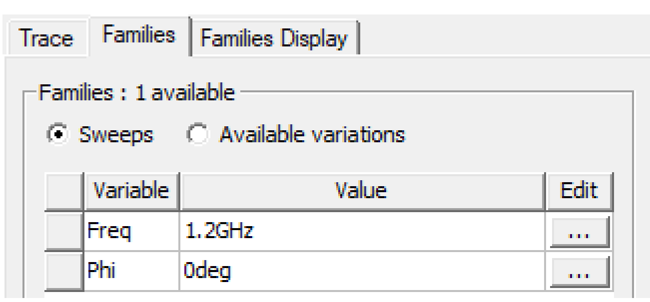


Figure 3-26: Spectral Radiation Plot Settings – *Families* Tab

9. Click **New Report** and then **Close**.

   The resulting plot should match the figure below:

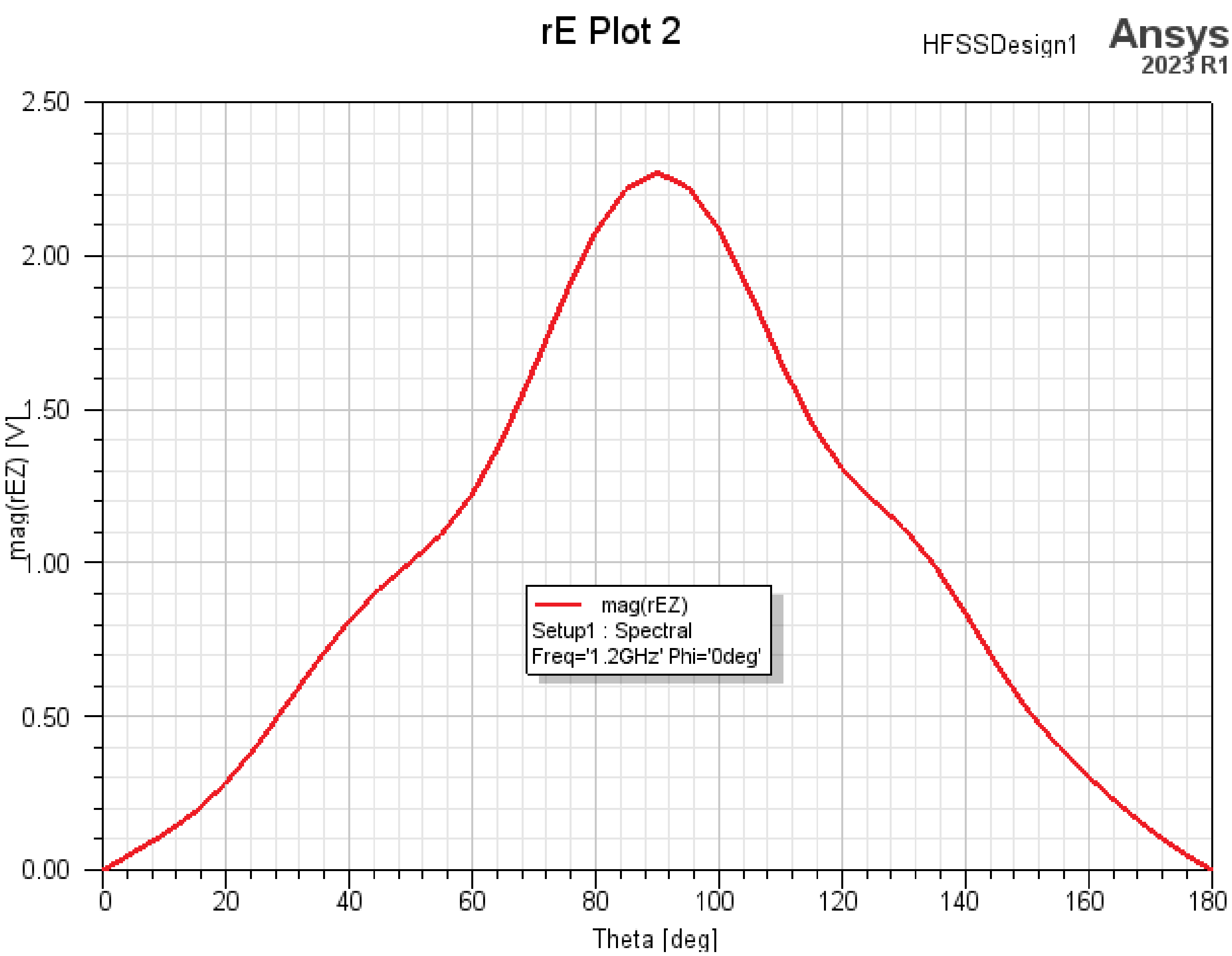


Figure 3-27: Spectral Radiation Plot

10. **Save** the project.

4 - Optionally, Restore Current View Orientations

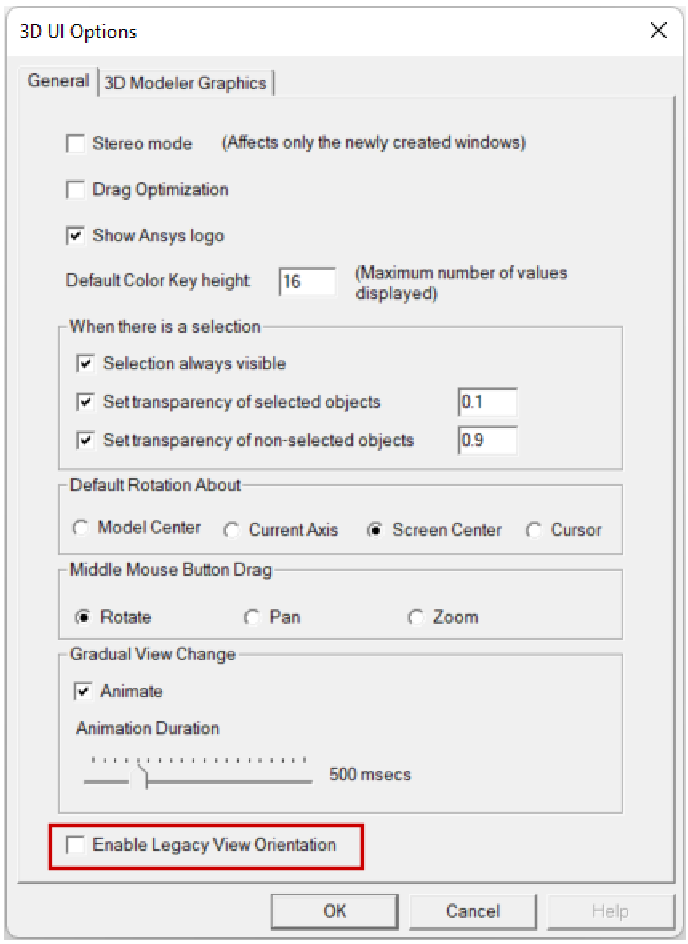
You have completed this getting started guide.

If you prefer to use the new view orientations implemented in version 2024 R1 of the Ansys Electronics Desktop application, clear the *Use Legacy View Orientation* option as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:



3. Click **OK**.

The settings in the 3D UI Options dialog box are global. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

You can now save and close this project.