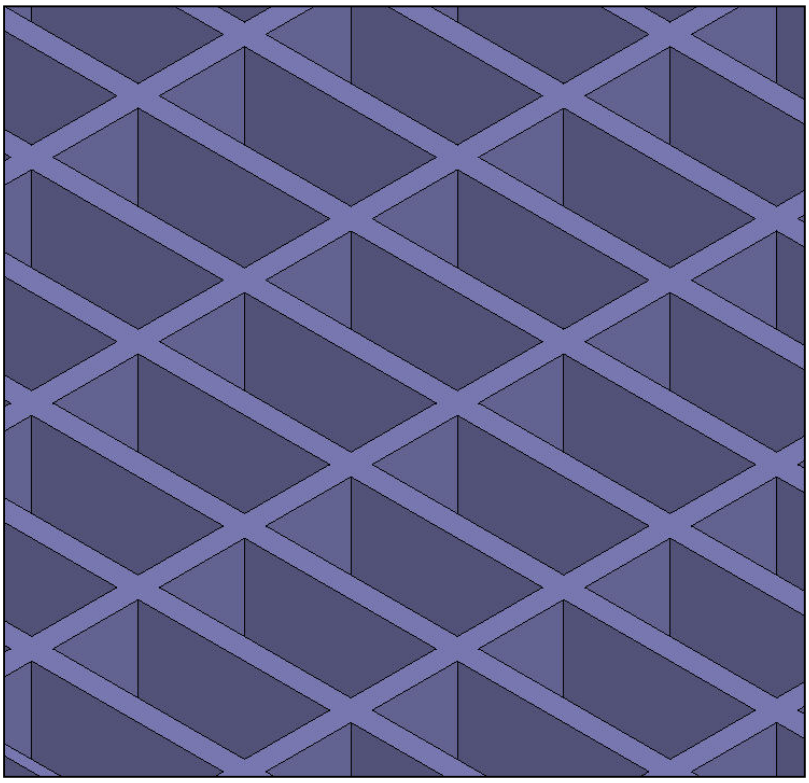
Example - Endfire Antenna Array

The Endfire Waveguide Array

- This example is intended to show you how to create, simulate, and analyze a Waveguide array antenna using the Ansoft HFSS Design Environment
- A WavePort excitation will be used for the feed
- PMLs (Perfectly Matched Layers) will be used for the radiation load
- Master/Slave boundary conditions will be used to create the array



Reference:

- [1] C.A. Balanis, “Antenna Theory – Analysis and Design”, Harper and Row, Publishers, Inc., 1982, ISBN 0-06-040458-2, section 6.2.
- [2] S.W. Lee and W.R. Jones, “On the Suppression of Radiation Nulls and Broadband Impedance Matching of Rectangular Waveguide Phased Arrays”, IEEE Trans. on Antennas Propagat., vol. AP-19, No. 1, pp. 41-51, Jan. 1971.

Design Review

1. Instead of modeling the entire array, we will make use of the master/slave boundary conditions and only model a unit cell.
2. Since the Master/Slave boundary conditions also allow us to change the scan angle, we will need a better radiation load on the top of the antenna than a simple radiation boundary condition. We will need to use a PML.

Ansoft HFSS Design Environment

- The following features of the Ansoft HFSS Design Environment are used to create this passive device model
  - 3D Solid Modeling
    - Primitives: **Box**
    - Other: **Perfectly Matched Layer (PML)**
  - Boundaries/Excitations
    - Ports: **Wave Ports**
    - Boundaries: **Master/Slave, Impedance**
  - Analysis
    - Sweep**: None**
  - Results
    - **Cartesian plotting**
  - Field Overlays:
    - **Radiation Patterns**
  - Optimetrics
    - **Parametric sweep**

Getting Started

Launching Ansoft HFSS

1. To access Ansoft HFSS, click the Microsoft **Start** button, select **Programs**, and select the **Ansoft > HFSS 10** program group. Click **HFSS 10**

Setting Tool Options

To set the tool options:

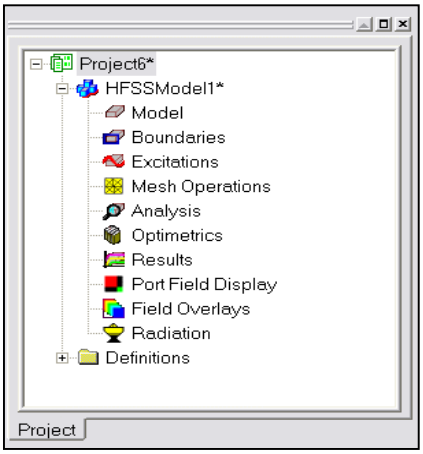
Note: In order to follow the steps outlined in this example, verify that the following tool options are set :

1. Select the menu item *Tools > Options > HFSS Options*
2. HFSS Options Window:
   1. Click the **General** tab
      - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
      - Duplicate boundaries with geometry: ☑ **Checked**
   2. Click the **OK** button
3. Select the menu item *Tools > Options > 3D Modeler Options*.
4. 3D Modeler Options Window:
   1. Click the **Operation** tab
      - Automatically cover closed polylines: ☑ **Checked**
   2. Click the **Drawing** tab
      - Edit property of new primitives: ☑ **Checked**
   3. Click the **OK** button

Opening a New Project

To open a new project:

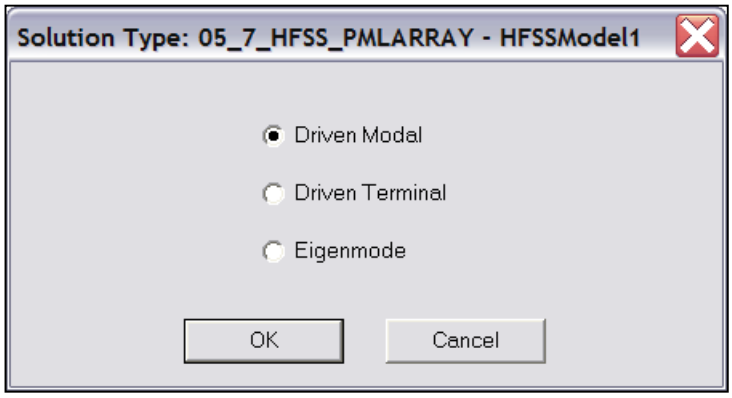
1. In an Ansoft HFSS window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
2. From the *Project* menu, select *Insert HFSS Design*.



Set Solution Type

To set the solution type:

1. Select the menu item *HFSS > Solution Type*
2. Solution Type Window:
   1. Choose **Driven Modal**
   2. Click the **OK** button

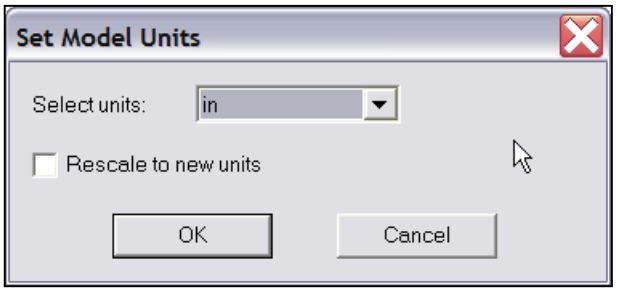


Creating the 3D Model

Set Model Units

To set the units:

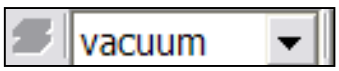
1. Select the menu item *3D Modeler > Units*
2. Set Model Units:
   1. Select Units: **in**
   2. Click the **OK** button



Set Default Material

To set the default material:

1. Using the 3D Modeler Materials toolbar, make sure that vacuum is the default material



Create Waveguide

To create the waveguide:

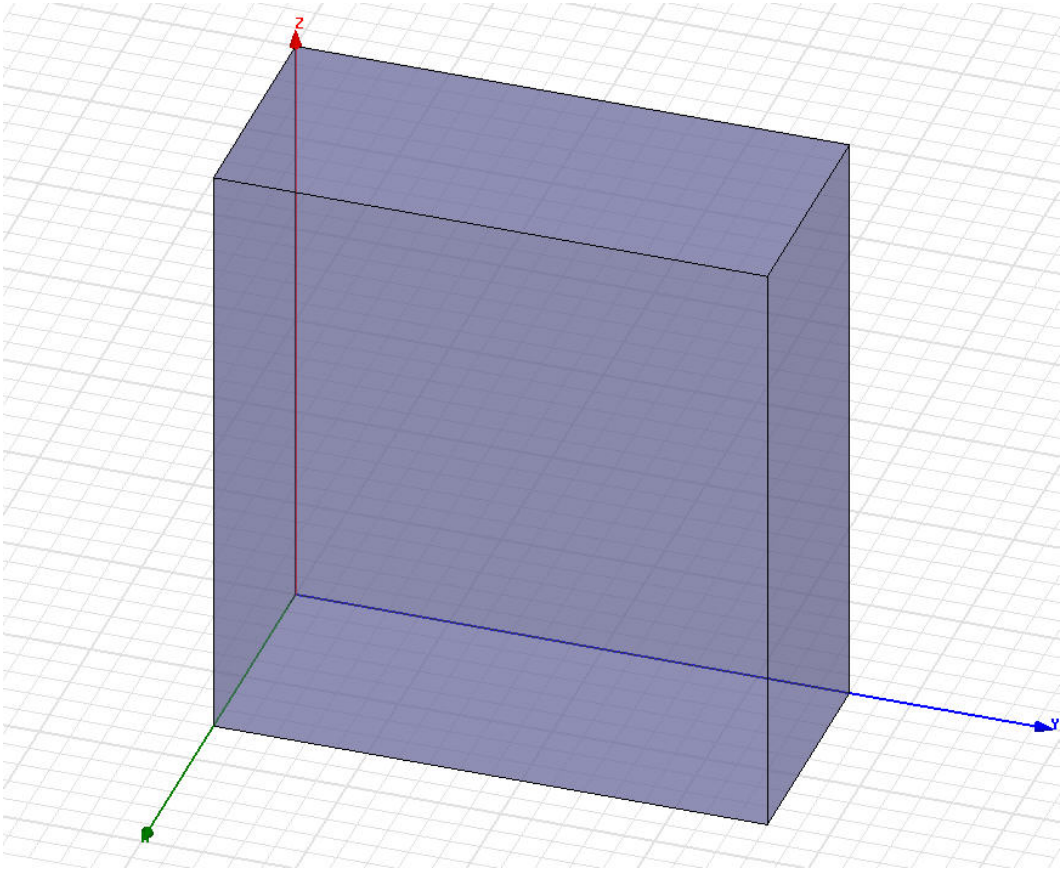
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box
   - dX: **0.4**, dY: **0.9**, dZ: **1.0**, Press the **Enter** key

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **waveguide**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View* or press the **CTRL+D** keys



Create Airbox

To create the airbox:

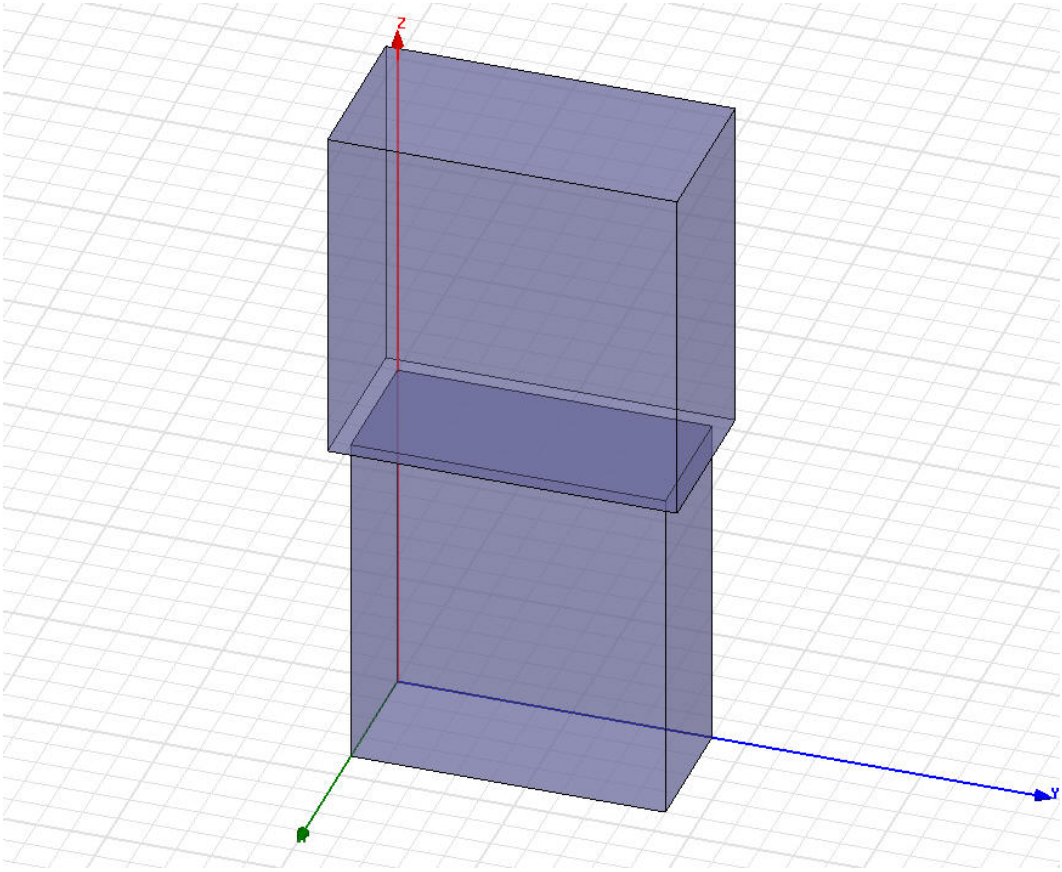
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **-0.05**, Y: **-0.05**, Z: **1.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box
   - dX: **0.5**, dY: **1.0**, dZ: **1.0**, Press the **Enter** key

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **airbox**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View*



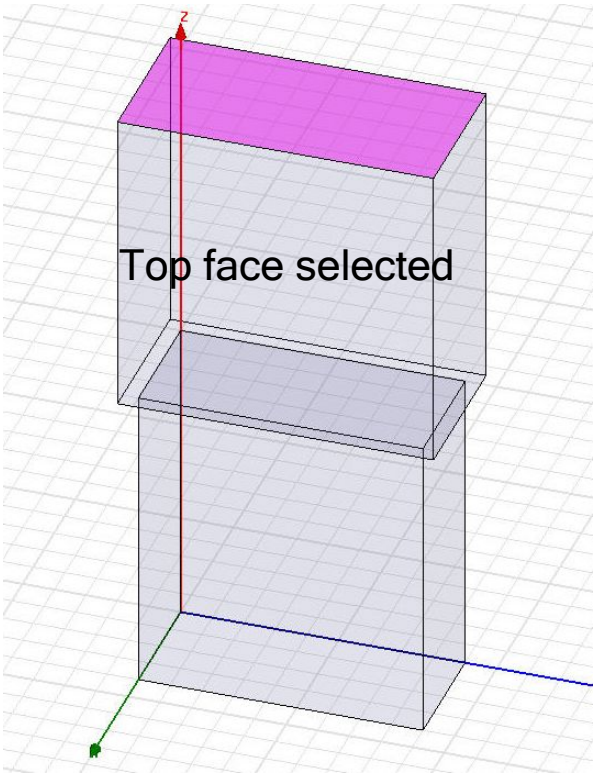
Create PML load

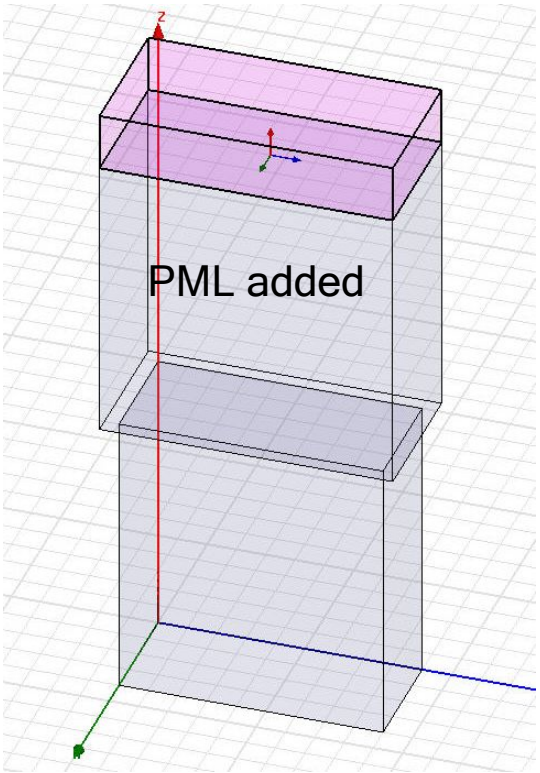
To select the proper face of the airbox:

1. Select the menu item *Edit > Select > Faces*
2. Graphically pick the top face of the airbox that was just created.

To assign the PML boundary

1. Select the menu item *HFSS > Boundaries > PML Setup Wizard*
2. PML Setup Wizard: Cover Objects
    1. Select: **Create PML Cover Objects on Selected Faces**
    2. Uniform Layer Thickness: **0.2in**
    3. Click the **Next** button
3. PML Setup Wizard: Material Parameters
    1. Select: **PML Objects accept Free Radiation**
        1. Min Frequency**: 9 GHz**
        2. Minimum Radiating Distance**: 1in**
        3. Click the **Next** button
    2. Review settings on the **PML Setup Wizard: Summary page**
    3. Click the **Finish** button





Make the PML object visible

By default, the PML wizard will turn off visibility of the PML object once it is created. We want to turn it back on again.

1. Select the menu item *View > Active View Visiblility*
2. Check the box next to **PML_airbox1**
3. Click *Done*

Create the Master/Slave boundary objects

Set Grid Plane

To set the grid plane:

1. Select the menu item *3D Modeler > Grid Plane > YZ*

Draw the Master / Slave objects

To draw the first Master/Slave rectangle:

1. Select the menu item *Draw > Rectangle*
2. Using the coordinate entry fields, enter the box position
   - X: **0.45**, Y: **-0.05**, Z: **1.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the rectangle
   - dX: **0**, dY: **1.0**, dZ: **1.2**, Press the **Enter** key

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **master1**
3. Click the **OK** button

To set the grid plane:

1. Select the menu item *3D Modeler > Grid Plane > XZ*

To draw the second Master/Slave rectangle:

1. Select the menu item *Draw > Rectangle*
2. Using the coordinate entry fields, enter the box position
   - X: **-0.05**, Y: **-0.05**, Z: **1.0**, Press the **Enter** key
2. Using the coordinate entry fields, enter the opposite corner of the rectangle
   - dX: **0.5**, dY: **0**, dZ: **1.2**, Press the **Enter** key

Create the Master/Slave boundary objects (continued)

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **master2**
3. Click the **OK** button

To duplicate the objects to create the slaves

1. Select the menu item *Edit > Select > Objects*
2. Select the menu item *Edit > Select > By Name*
3. Select the object, **master1**
4. Select the menu item *Edit > Duplicate > Along Line*
5. Using the coordinate entry fields, enter the start position of the duplicate vector
   - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
6. Using the coordinate entry fields, enter the end point of the duplicate vector
   - dX: **-0.5**, dY: **0**, dZ: **0**, Press the **Enter** key
7. When the dialog box pops up requesting the total number of copies, change the value to **2**, Press the **OK** button.
8. Repeat this process for the object **master2** using a duplicate vector of **<0,1,0>**

Change Slave boundary names

To change the duplicated master boundary to slave boundary

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
   1. Select the objects named: **master1_1**
   2. Click the **OK** button
3. Select the menu Item: *Edit > Properties*
4. Change the name to: **slave1**
5. Repeat the process for **master2_1 → slave2**

Assign Master/Slave Boundaries

- **To create a Master boundary**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **master1**
     2. Click the **OK** button
  3. Select the menu item *HFSS > Boundaries > Assign > Master*
  4. Master Boundary window
     1. Name: **master1**
     2. Coordinate System: U Vector: click **Undefined** pulldown
     3. Using the coordinate entry fields, enter the start position
        - X:**0.45**, Y: **-0.05,** Z:**1.0**, Press the **Enter** key
     4. Using the coordinate entry fields, enter the stop position of the vector
        - dX: **0**, dY: **1**, dZ: **0**, Press the **Enter** key
     5. For the V vector, check the **Reverse Direction** box
- **Repeat the process for master2 using the following points:**
  1. Using the coordinate entry fields, enter the start position
     - X:**0.45**, Y: **-0.05,** Z:**1.0**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the stop position of the vector
     - dX:**-0.5**, dY: **0**, dZ: **0**, Press the **Enter** key
- **To create a Slave boundary**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **slave1**
     2. Click the **OK** button
  3. Select the menu item *HFSS > Boundaries > Assign > Slave*
  4. Slave Boundary window
     1. Name: **slave1**
     2. Master Boundary: click on **Undefined** pulldown and select **Master1**
     3. Coordinate System: U Vector: click **Undefined** pulldown
     4. Using the coordinate entry fields, enter the start position
        - X:**-0.05**, Y: **-0.05,** Z:**1.0**, Press the **Enter** key

Assign Master / Slave Boundaries (continued)

To create a Slave boundary (continued)

1. Using the coordinate entry fields, enter the stop position of the vector
   - dX: **0**, dY: **1**, dZ: **0**, Press the **Enter** key
2. Click the **Next** button
3. Make sure that **Use Scan Angles To Calculate Phase Delay** is checked
   1. For **Phi**, enter 0 deg
   2. For **Theta**, enter a variable name **theta_scan**, and hit **Enter**
   3. For the **Add Variable** dialog, enter **30deg** for **theta_scan**

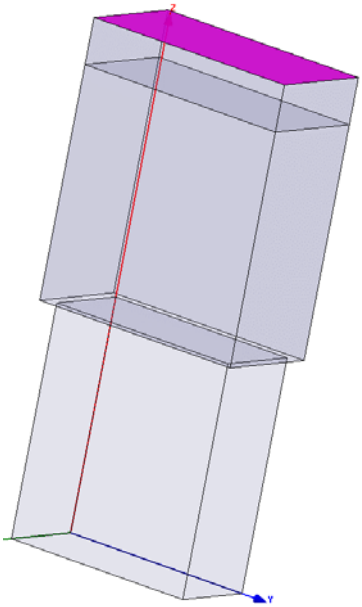
Repeat the process for slave2:

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
   1. Select the objects named: **slave2**
   2. Click the **OK** button
3. Select the menu item *HFSS > Boundaries > Assign > Slave*
4. Slave Boundary window
   1. Name: **slave2**
   2. Master Boundary: click on **Undefined** pulldown and select **Master2**
   3. Coordinate System: U Vector: click **Undefined** pulldown
   4. Using the coordinate entry fields, enter the start position
      - X:**0.45**, Y: **0.95,** Z:**1.0**, Press the **Enter** key
   5. Using the coordinate entry fields, enter the stop position of the vector
      - dX:**-0.5**, dY: **0**, dZ: **0**, Press the **Enter** key

Assign Impedance Boundary

To create the impedance boundary on the PML object:

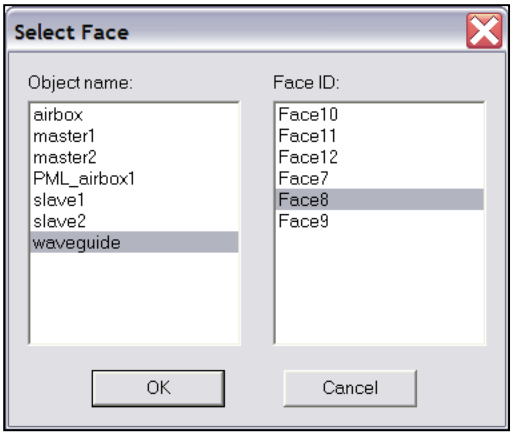
1. Select the menu item *Edit > Select > Faces*
2. Graphically pick the top face of the PML object
3. Select the menu item: *HFSS > Boundaries > Assign > Impedance*
4. Name: **TopLoad**
5. Resistance: **377*cos(theta_scan)**
6. Reactance: **0**

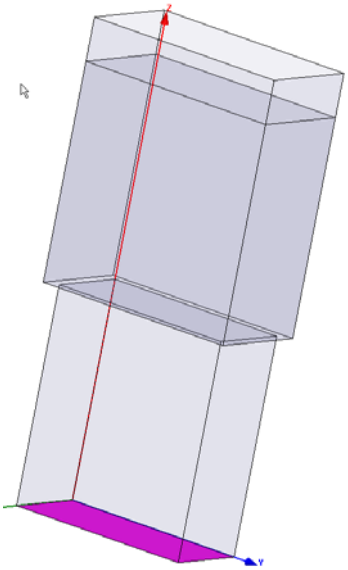


Create WavePort

To assign waveport to waveguide object:

1. Select the menu item *Edit > Select > By Name*
2. Select Face dialog: Select the object **waveguide** from the left column
3. Select different FaceIDs until the bottom face of the waveguide is highlighted.
4. Click **OK**





Create WavePort (continued)

To assign waveport to waveguide object (continued):

1. Select the menu item *HFSS > Excitation > Assign > WavePort*
2. Name: **p1**
3. Click **Next**
4. Wave Port: Modes
    1. Click **Next**
5. Wave Port: Post-Processing
    1. Click **Finish**

Analysis Setup

Creating an Analysis Setup

- **To create an analysis setup:**
  1. Select the menu item *HFSS > Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **General** tab:
        - Solution Frequency: **9.25 GHz**
        - Maximum Number of Passes: **5**
        - Maximum Delta S: **0.0001**
     2. Click the **OK** button

Create a Radiation Setup

- **To define the radiation setup**
  1. Select the menu item *HFSS > Radiation > Insert Far Field Setup > Infinite Sphere*
  2. Far Field Radiation Sphere Setup dialog
     1. Select the **Infinite Sphere** Tab
        1. Name: **ff_all**
        2. Phi: (Start: **0**, Stop: **0**, Step Size: **10**)
        3. Theta: (Start: **0**, Stop: **360**, Step Size: **2**)
     2. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft HFSS window, select the menu item *File > Save As*.
  2. From the **Save As** window, type the Filename: **hfss_pmlarray**
  3. Click the **Save** button

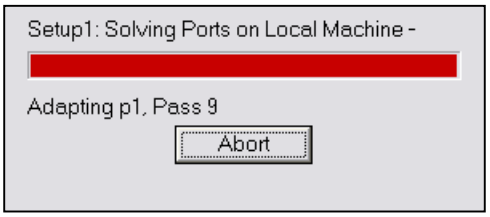
Analyze

Model Validation

- **To validate the model:**
  1. Select the menu item *HFSS > Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item *HFSS > Analyze All*



Solution Data

To view the Solution Data:

1. Select the menu item *HFSS > Results > Solution Data*

   To view the Profile:

   1. Click the **Profile** Tab.

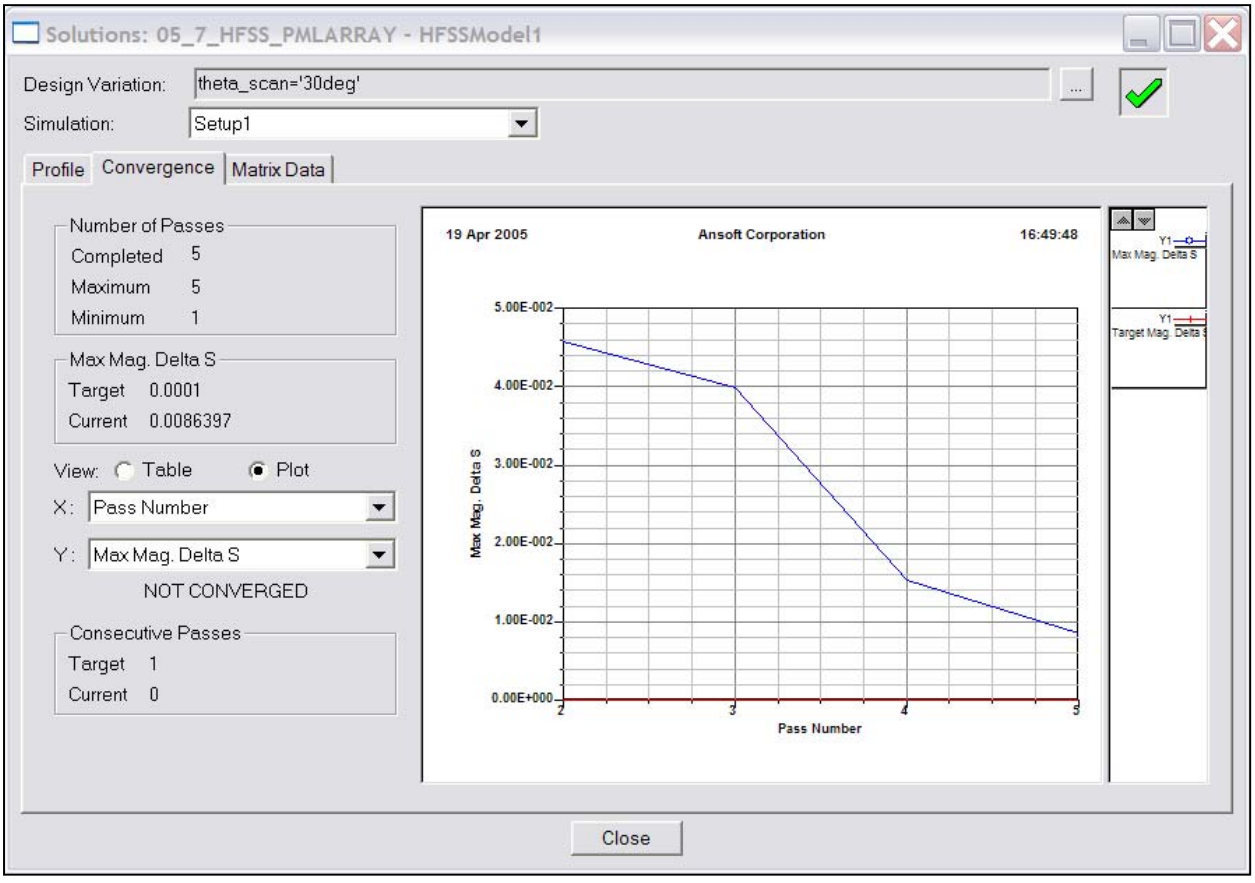
   To view the Convergence:

   1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

   To view the Matrix Data:

   1. Click the **Matrix Data** Tab
      - **Note:** To view a real-time update of the Matrix Data, set the Simulation to **Setup1, Last Adaptive** during solution.

2. Click the **Close** button when done viewing or simulation complete.

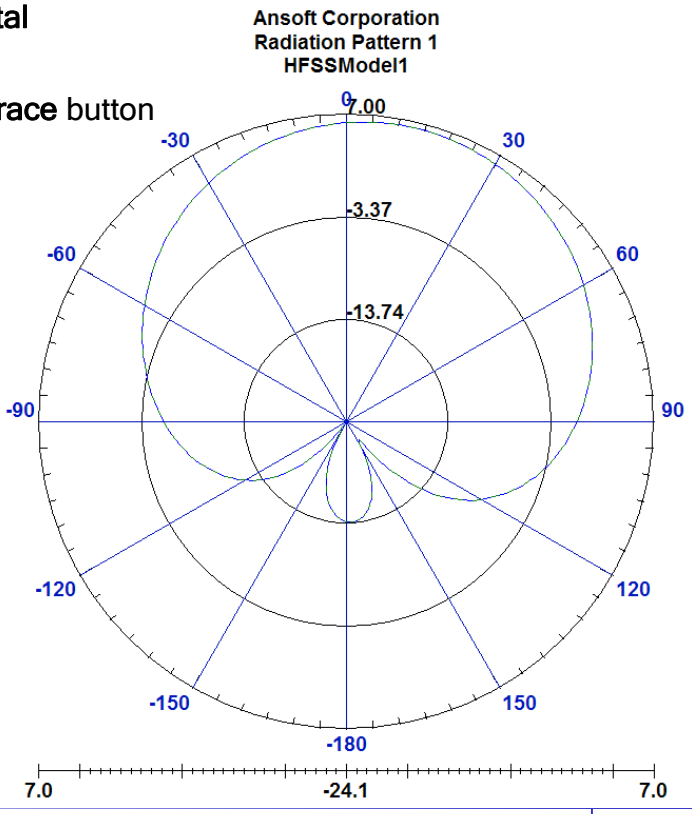


Far Field Plots

Create Far Field Plot

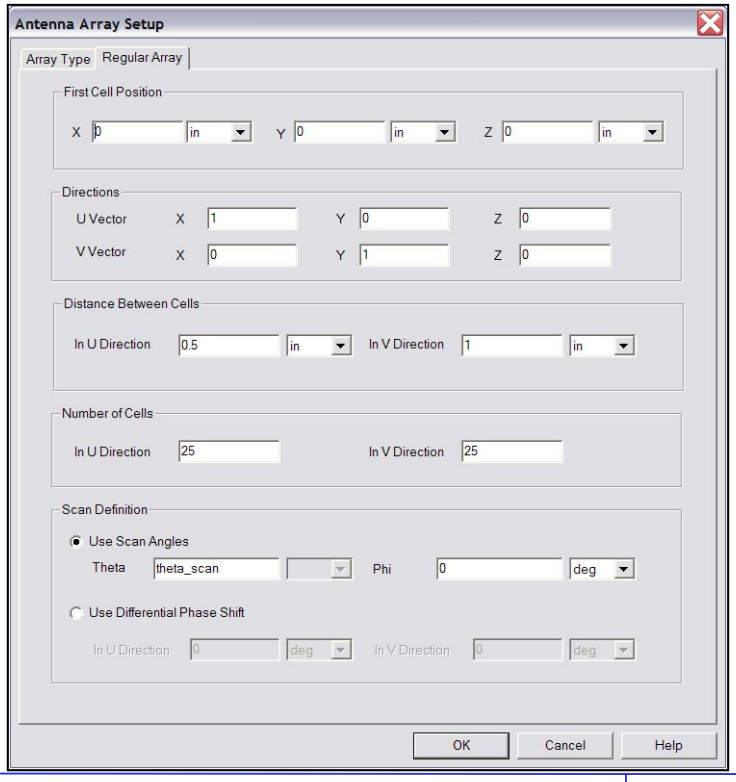
To create a 2D polar far field plot :

1. Select the menu item *HFSS > Results > Create Report*
2. Create Report Window:
    1. Report Type: **Far Fields**
    2. Display Type: **Radiation Pattern**
    3. Click the **OK** button
3. Traces Window:
    1. Solution: **Setup1: LastAdaptive**
    2. Geometry: **ff_all**
    3. In the **Sweeps** tab:
        1. Select **Phi** under the **Name** column, and on the drop list, select **Theta**. This changes the primary sweep to Theta.
    4. In the **Mag** tab
        1. Category: **Directivity**
        2. Quantity: **DirTotal**
        3. Function: **dB**
        4. Click the **Add Trace** button
    5. Click the **Done** button

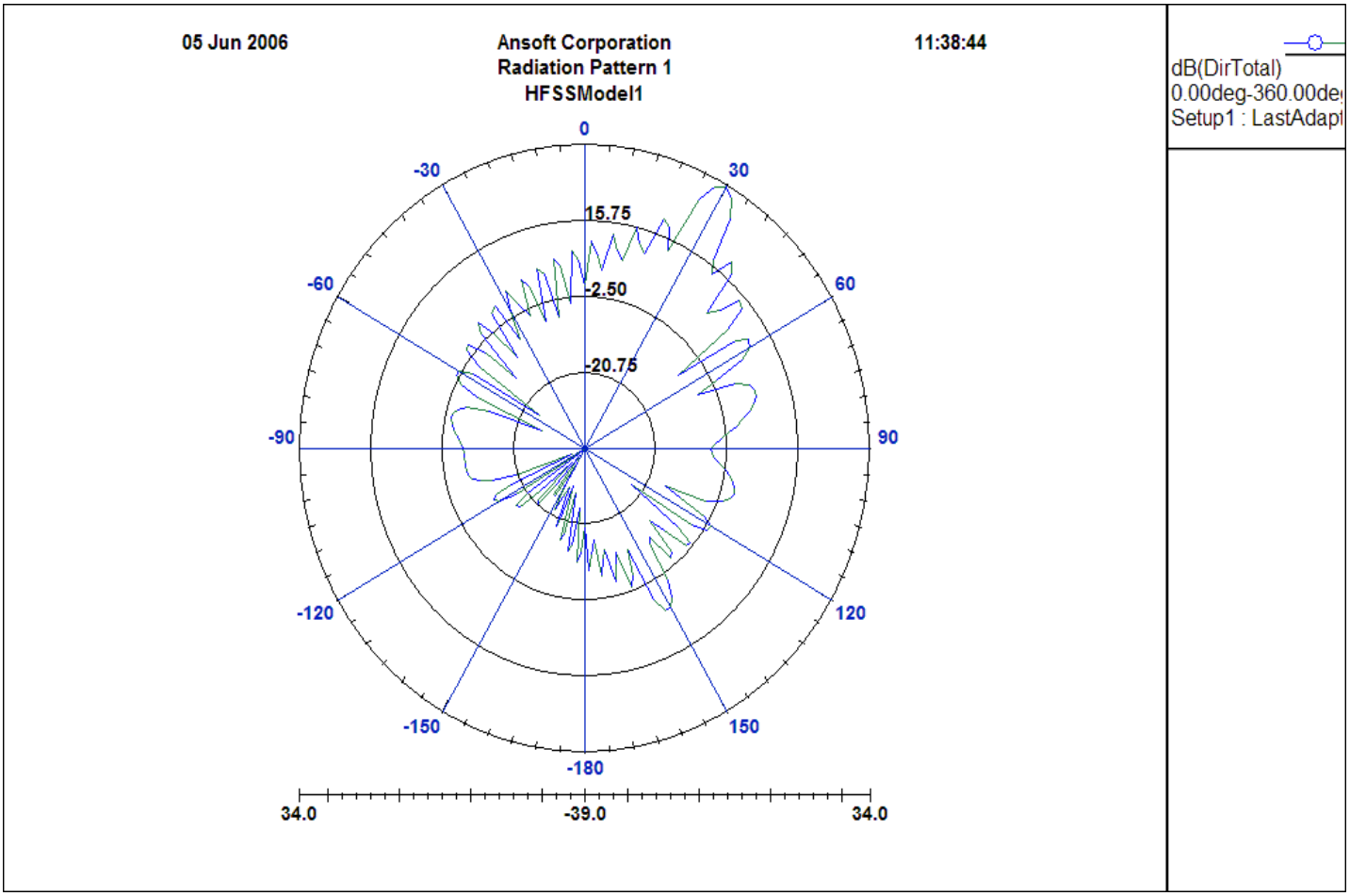


Add Array Factor to Plot

- **To setup array factor:**
  1. Select the menu item *HFSS > Radiation > Antenna Array Setup*
  2. Select the radio button for **Regular Array Setup**
  3. Switch to the **Regular Array** tab
  4. First Cell Position:
     - X:**0in**, Y: **0 in**, Z:**0 in**
  5. Directions: U vector
     - X:**1**, Y: **0**, Z:**0**
  6. Directions: V Vector
     - X:**0**, Y: **1**, Z:**0**
  7. Distance Between Cells
     - U Direction: **0.5 in**
     - V Direction **1 in**
  8. Number of Cells
     - U Direction: **25**
     - V Direction **25**
  9. Scan Definition: Use Scan Angles
     - Theta: **theta_scan**
     - Phi: **0 deg**



Add Array Factor to Plot (continued)



Optimetrics Setup - Parametric Sweep

- For this array design, we want to see the effect of scan angle on the input match of the antenna. To do this, we must sweep the scan angle with a parametric sweep.

Add a Parametric Sweep

1. Select the menu item *HFSS > Optimetrics Analysis > Add Parametric*
2. Setup **Sweep Analysis** Window:
    1. Click the **Sweep Definitions** tab:
        1. Click the **Add** button
        2. Add/Edit Sweep Dialog
            1. Select Variable: **theta_scan** (this is the only variable defined, so it is greyed out)
            2. Select **Linear Step**
            3. Start: **0deg**
            4. Stop: **60deg**
            5. Step: **10deg**
            6. Click the **Add** button
            7. Click the **OK** button

Analyze Parametric Sweep

- **To start the solution process:**
    1. Expand the Project Tree to display the items listed under **Optimetrics**
    2. Right-click the mouse on **ParametricSetup1** and choose **Analyze**

Optimetrics Results

- **To view the Optimetrics Results:**
    1. Select the menu item *HFSS > Optimetrics Analysis > Optimetrics Results*
    2. Select the **Profile** Tab to view the solution progress for each setup.
    3. Click the **Close** button when you are finished viewing the results

Create S-Parameter Plot - S11 at each θ

To create a report:

1. Select the menu item *HFSS > Results > Create Report*
2. Create Report Window:
    1. Report Type: **Modal Solution Data**
    2. Display Type: **Rectangular Plot**
    3. Click the **OK** button
3. Traces Window:
    1. Solution: **Setup1: LastAdaptive**
    2. Click the **Sweeps** tab
        1. Select **Sweep Design and Project variable values**
    3. In the **Sweeps** tab:
        1. Select **Freq** under the **Name** column, and on the drop list, select **theta_scan**. This changes the primary sweep to theta_scan.
    4. Click the **Y** tab
        1. Category: **S Parameter**
        2. Quantity: **S(p1,p1)**
        3. Function: **dB**
        4. Click the **Add Trace** button
    5. Click the **Done** button