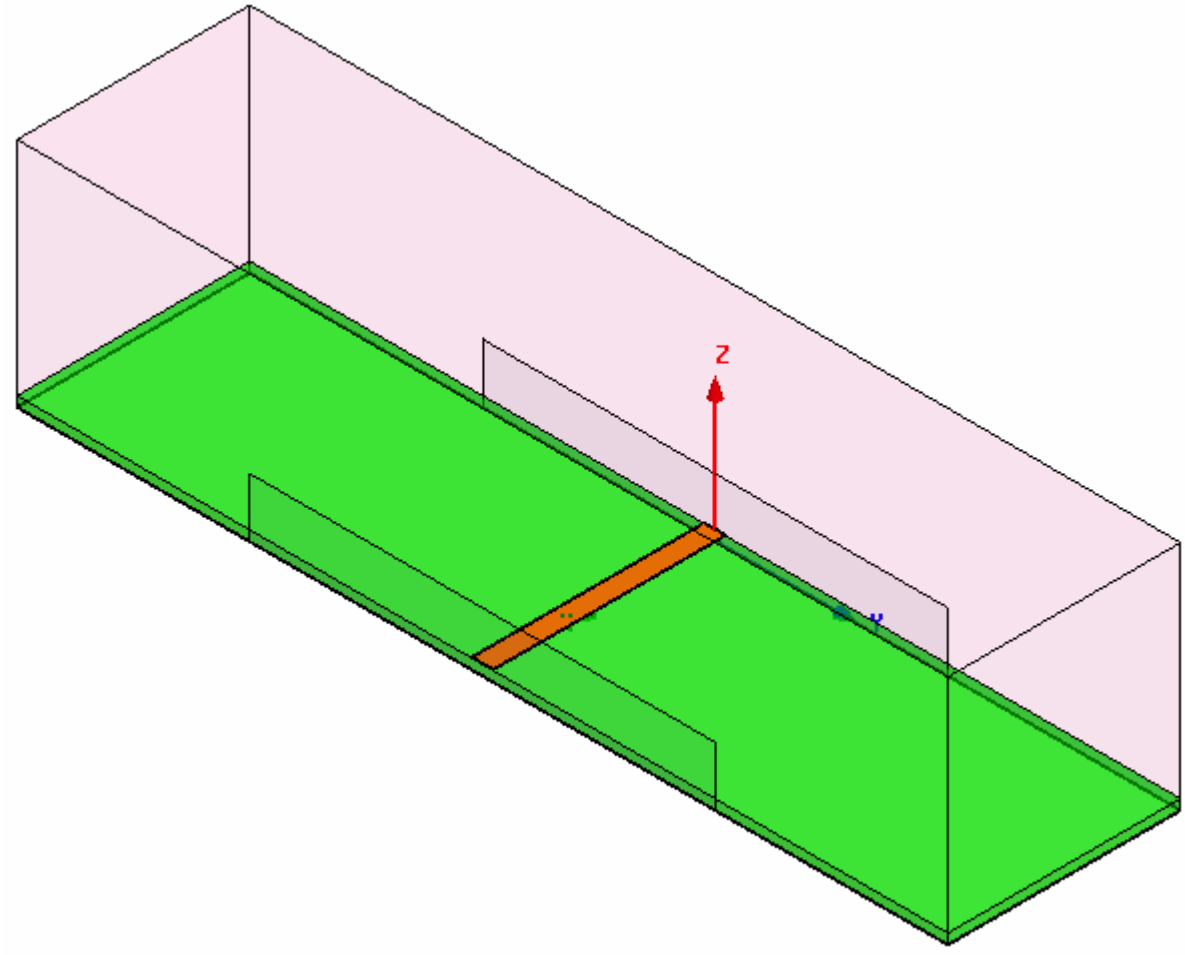
Microstrip Wave Port

- This example is intended to show you how wave port size can influence the results of any simulation using the Ansoft HFSS Version 9 Design Environment.



Ansoft Design Environment

- The following features of the Ansoft HFSS Design Environment are used to create this passive device model
  - 3D Solid Modeling
    - Primitives: **Boxes, Rectangle**
    - Boolean Operations: **Duplicate Along Line**
  - Boundary/Excitation
    - Ports: **Wave Ports and Integration Lines**
  - Analysis
    - Solution: **Ports Only**
    - Sweep: **Interpolating**
  - Results
    - **Cartesian plotting**
  - Fields Overlays
    - **Port Field Display**

Getting Started

Launching Ansoft HFSS

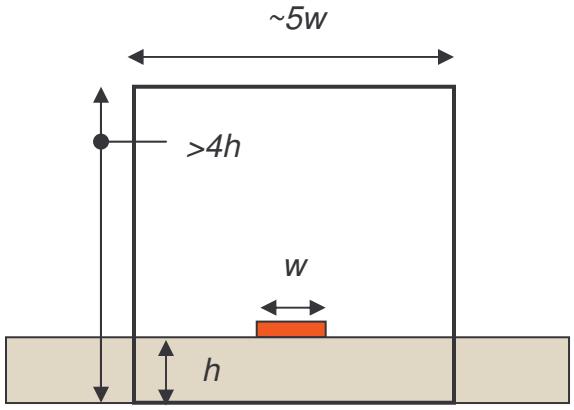
1. To access Ansoft HFSS, click the Microsoft **Start** button, select **Programs**, and select the **Ansoft, HFSS 9** program group. Click **HFSS 9**.

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

Design Review

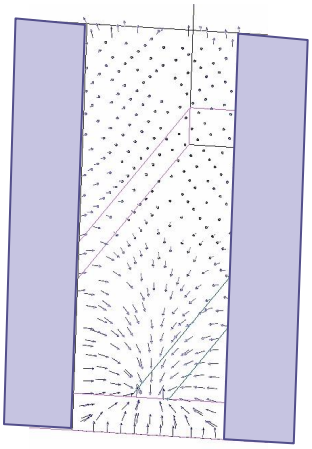
- Generally speaking when we assign a wave port to a microstrip line, or any quasi-TEM line, we need to include some area around the actual transmission line. The big question is, “how much area?”
- Below you will see the cross section of a simple microstrip line with naming conventions shown.



- As a rule of thumb, we typically create a 2D rectangle to represent the wave port stimulus for this type of structure. The dimensions of the rectangle are roughly five times the width of the transmission line by four times the height of the substrate.
- REMEMBER: These are only guidelines !!!
- The height of the port will be affected by the permittivity of the substrate. The higher the permittivity, the less the fields will propagate in the air, and the shorter the port can be made.

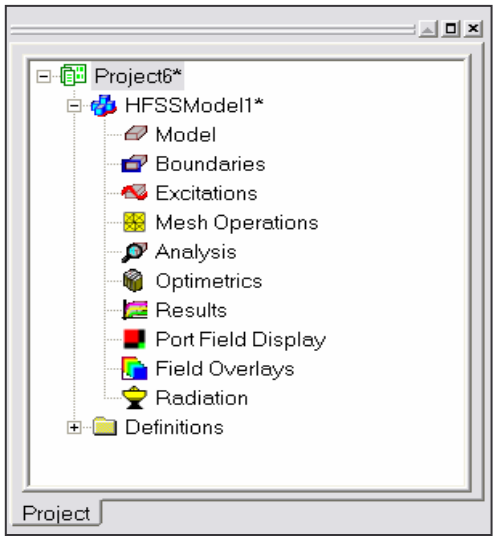
The width of the port will affect the port impedance and propagating modes. The narrower the width (image on right), the more the fields will couple to the side walls of the port. This effect may not be physical. The wider the port, the greater chance that a higher frequency waveguide mode can propagate.

This example will explore this last phenomenon.



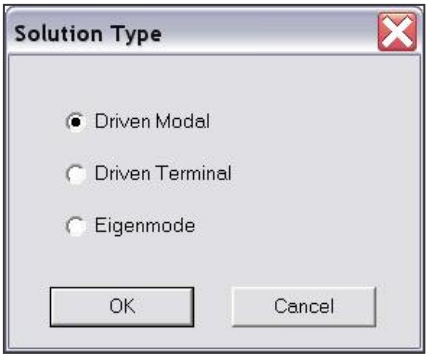
Opening a New Project

- To open a new project:
  - In an Ansoft HFSS window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - From the *Project* menu, select *Insert HFSS Design*.



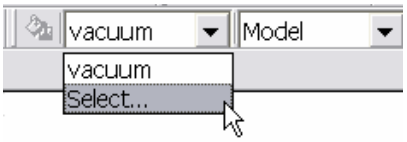
Set Solution Type

- To set the solution type:
  - Select the menu item *HFSS > Solution Type*
  - Solution Type Window:
    - Choose **Driven Modal**
    - Click the **OK** button



Set Model Units

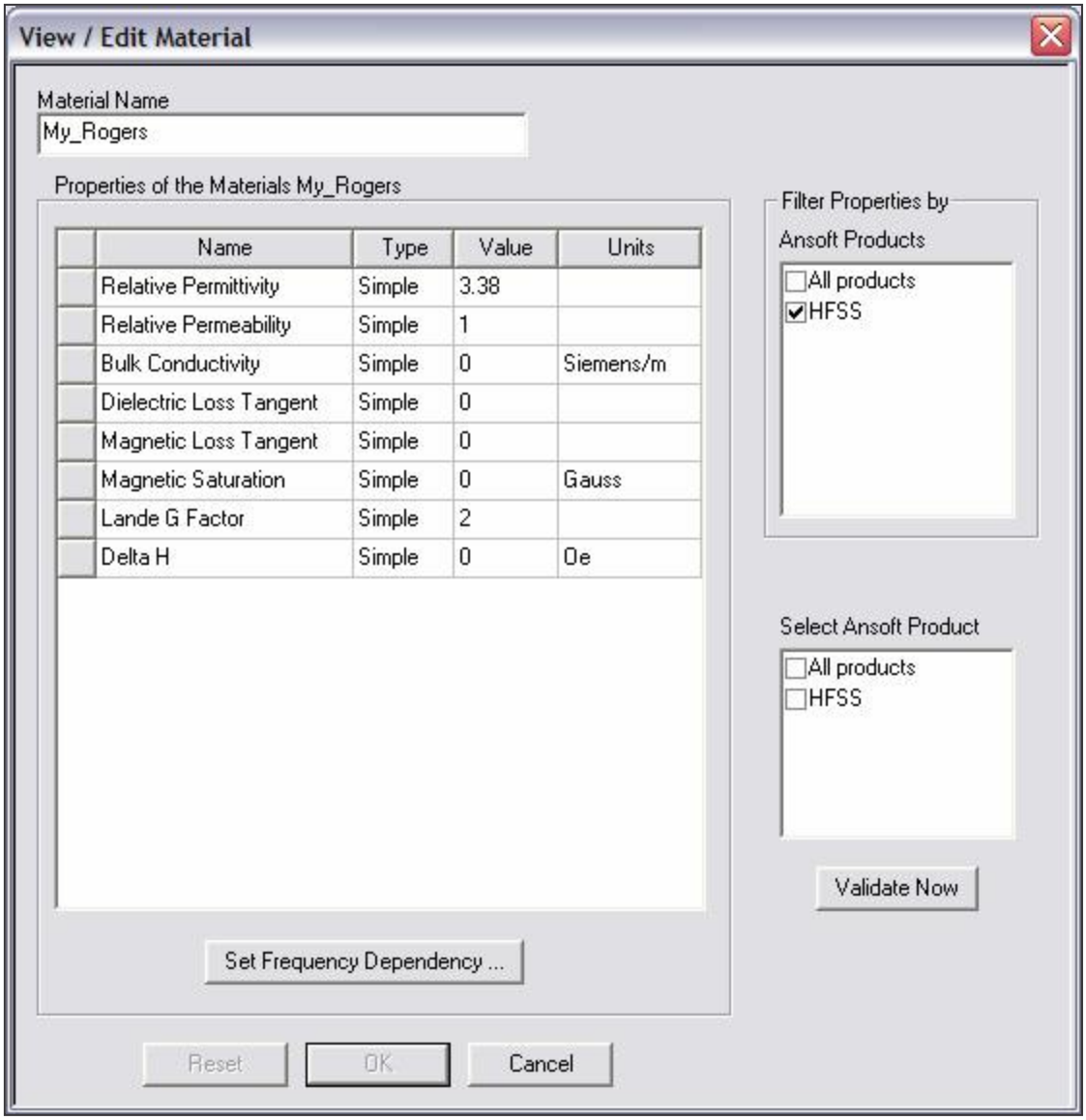
- To set the units:
  - Select the menu item *3D Modeler > Units*
  - Set Model Units:
    - Select Units: **mil**
    - Click the **OK** button



Add New Material

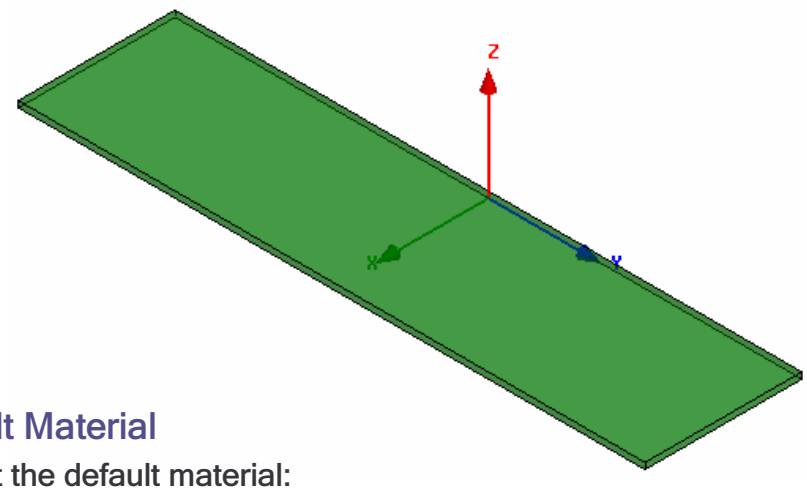
To add a new material:

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. From the Select Definition window, click the **Add Material** button
3. View/Edit Material Window:
    1. Material Name: **My_Rogers**
    2. Relative Permittivity: **3.38**
    3. Click the **OK** button
4. Click the **OK** button



Create Substrate

- To create the substrate:
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **0.0**, Y: **-400.0**, Z: **0.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **200.0**, dY: **800.0**, dZ: **8.0**, Press the **Enter** key
- To set the name:
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Substrate**
  3. Click the **OK** button
- To fit the view:
  1. Select the menu item *View > Fit All > Active View*.



Set Default Material

- To set the default material:
  1. Using the 3D Modeler Materials toolbar, choose **Select**
  2. Select Definition Window:
     1. Type **pec** in the **Search by Name** field
     2. Click the **OK** button

Create Trace

To create the trace:

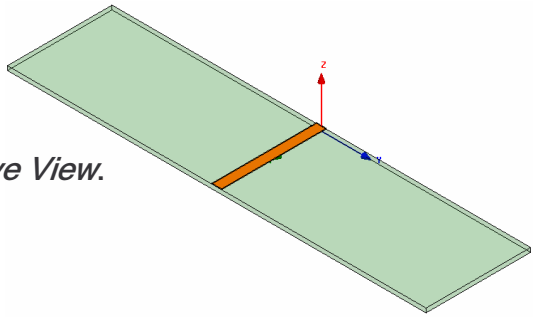
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **0.0**, Y: **-9.25**, Z: **8.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **200.0**, dY: **18.5**, dZ: **1.4**, Press the **Enter** key

To set the name:

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

To fit the view:

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



Create Ground

To create the ground:

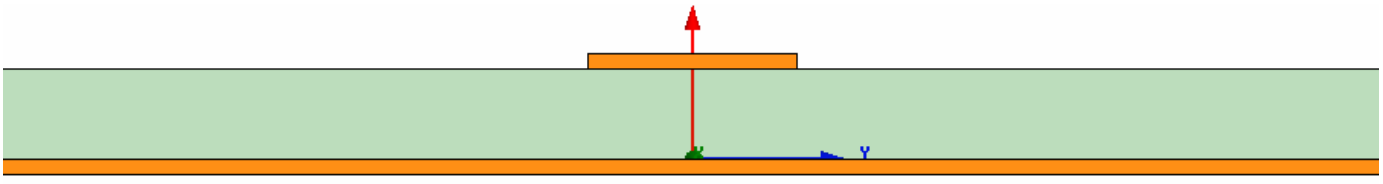
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **0.0**, Y: **-400.0**, Z: **0.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **200.0**, dY: **800.0**, dZ: **-1.4**, Press the **Enter** key

To set the name:

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

To fit the view:

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



Set Default Material

- To set the default material:
  1. Using the 3D Modeler Materials toolbar, choose **vacuum**

Create Air

- To create the Air:
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **0.0**, Y: **-400.0**, Z: **-1.4**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **200.0**, dY: **800.0**, dZ: **200.0**, Press the **Enter** key
- To set the name:
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Air**
  3. Click the **OK** button
- To fit the view:
  1. Select the menu item *View > Fit All > Active View*.

Create Radiation Boundary

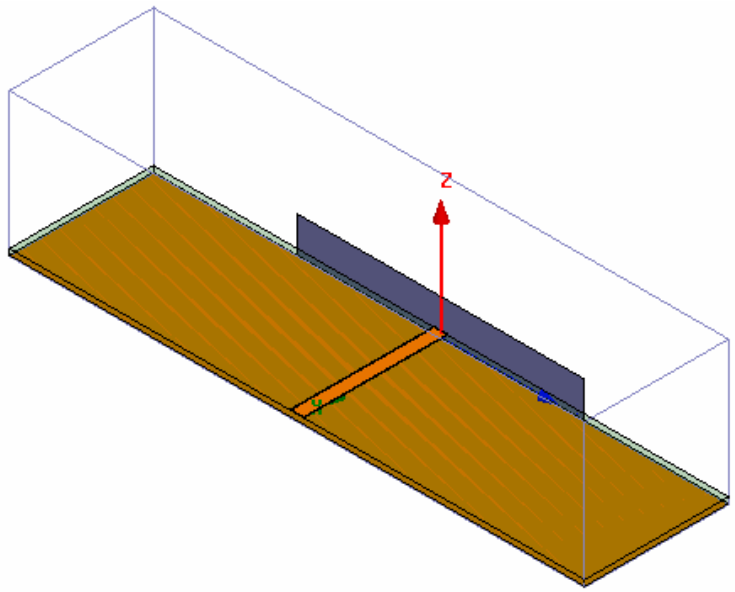
- To select the object Air:
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Air**
     2. Click the **OK** button
- To create a radiation boundary
  1. Select the menu item *HFSS > Boundaries > Assign > Radiation*
  2. Radiation Boundary Window
     - Name: **Rad1**
     - Click the **OK** button

Set Grid Plane

- To set the grid plane:
  1. Select the menu item *3D Modeler > Grid Plane > YZ*

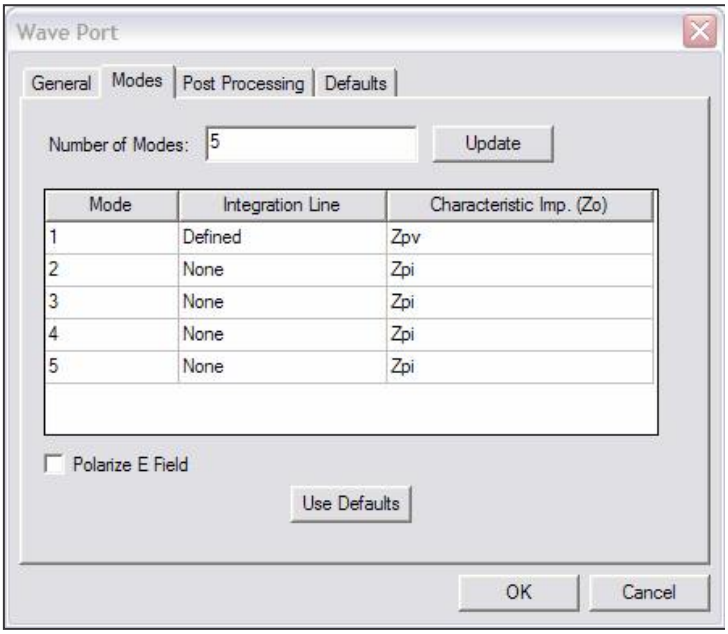
Create the Wave Port

- To create a rectangle that represents the port:
  1. Select the menu item *Draw > Rectangle*
  2. Using the coordinate entry fields, enter the center position
     - X: 0.0, Y: -200.0, Z: 0.0, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the rectangle:
     - dX: 0.0, dY: 400.0, dZ: 50.0, Press the **Enter** key
- To parameterize the object:
  1. Select the **Command** tab from the **Properties** window
  2. For **Position**, type: **0mil, -Port_Width/2, 0mil**, Click the **Tab** key to accept
     - Add Variable **Port_Width**: **400mil**, Click the **OK** button
  3. For **YSize**, type: **Port_Width**, Click the **Tab** key to accept
- To set the name:
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Port1**
  3. Click the **OK** button



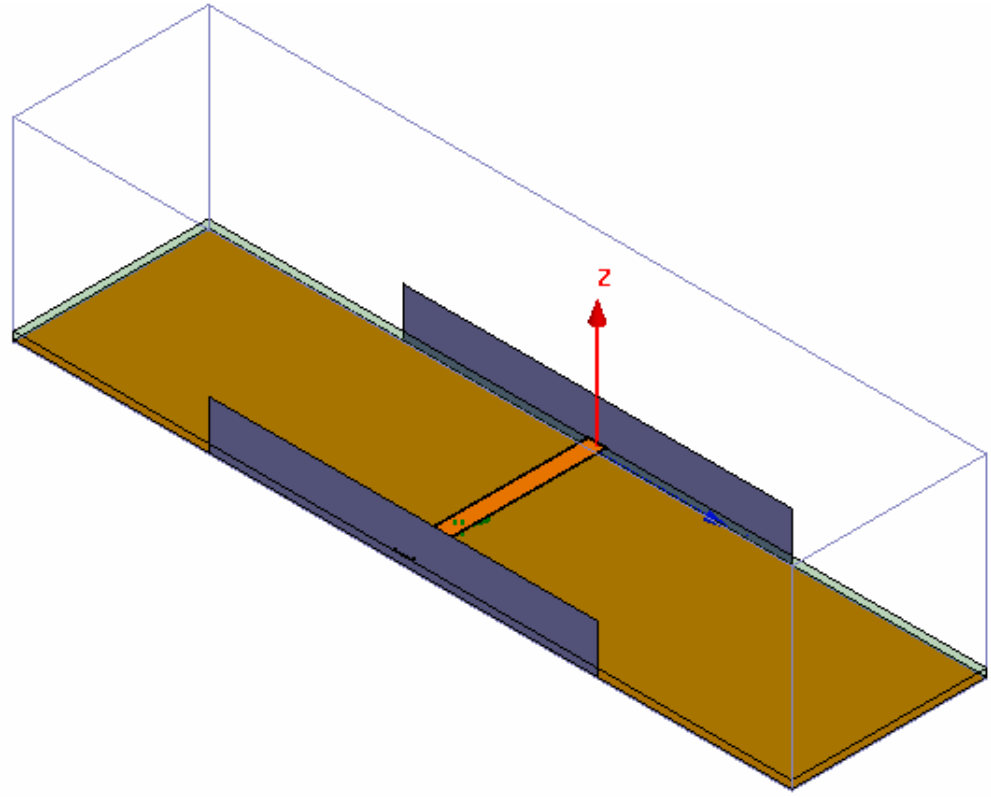
Assign the Excitation 1

- To select the object p1:
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Port1**
     2. Click the **OK** button
- To assign wave port excitation
  1. Select the menu item *HFSS > Excitations > Assign > Wave Port*
  2. Wave Port : General
     1. Name: **p1**,
     2. Click the **Next** button
  3. Wave Port : Modes
     1. Number of Modes: **5**, Click the **Update** button
     2. For Mode **1**, click the **Undefined** column and select **New Line**
     3. Using the coordinate entry fields, enter the vector position
        - X: **0.0**, Y: **0.0**, Z: **0.0,** Press the **Enter** key
     4. Using the coordinate entry fields, enter the vertex
        - dX: **0.0**, **dY: 0.0**, dZ: **8.0,** Press the **Enter** key
     - For Mode **1**, click the **Zpi** column and select **Zpv**
     1. Click the **Next** button
  4. Wave Port : Post Processing
     1. Click the **Next** button
  5. Click the **Finish** button



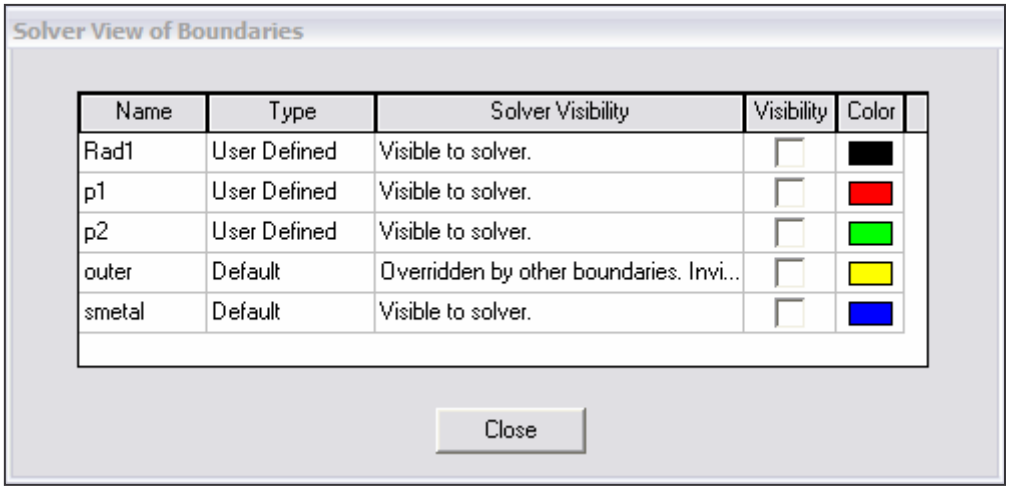
Create Wave Port Excitation 2

- To select the object Port1:
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Port1**
     2. Click the **OK** button
- To duplicate the port:
  1. Select the menu item, *Edit > Duplicate > Along Line*
  2. Using the coordinate entry fields, enter the first point of the duplicate vector
     - X: **0.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the second point of the duplicate vector
     - dX: **200.0**, dY: **0.0**, dZ: **0.0,** Press the **Enter** key
  4. Duplicate Along Line Windows
     1. Total Number: **2**
     2. Click the **OK** button
  5. Click the **Done** button



Boundary Display

- To verify the boundary setup:
  1. Select the menu item *HFSS > Boundary Display (Solver View)*
  2. From the Solver View of Boundaries, toggle the Visibility check box for the boundaries you wish to display.
     - **Note:** The background (Perfect Conductor) is displayed as the **outer** boundary.
     - **Note:** The Perfect Conductors are displayed as the **smetal** boundary.
     - **Note:** Select the menu item, *View > Visibility* to hide all of the geometry objects. This makes it easier to see the boundary
  3. Click the **Close** button when you are finished



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: **20GHz**
     2. Check **Solve Ports Only** ☑
     3. Click the **OK** button

Adding a Frequency Sweep

- To add a frequency sweep:
  1. Select the menu item *HFSS > Analysis Setup > Add Sweep*
     1. Select Solution Setup: **Setup1**
     2. Click the **OK** button
  2. Edit Sweep Window:
     1. Sweep Type: **Interpolating**
        - Error Tolerance: **0.5%**
        - Max Solutions: **20**
     2. Frequency Setup Type: **Linear Count**
        - Start: **0.1GHz**
        - Stop: **50.1GHz**
        - Count: **81**
     3. 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_waveports**
  3. Click the **Save** button

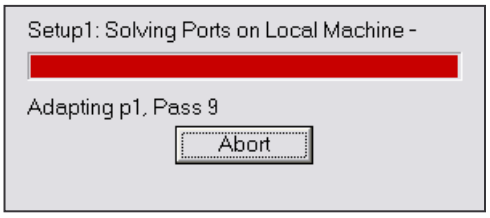
Analyze

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.

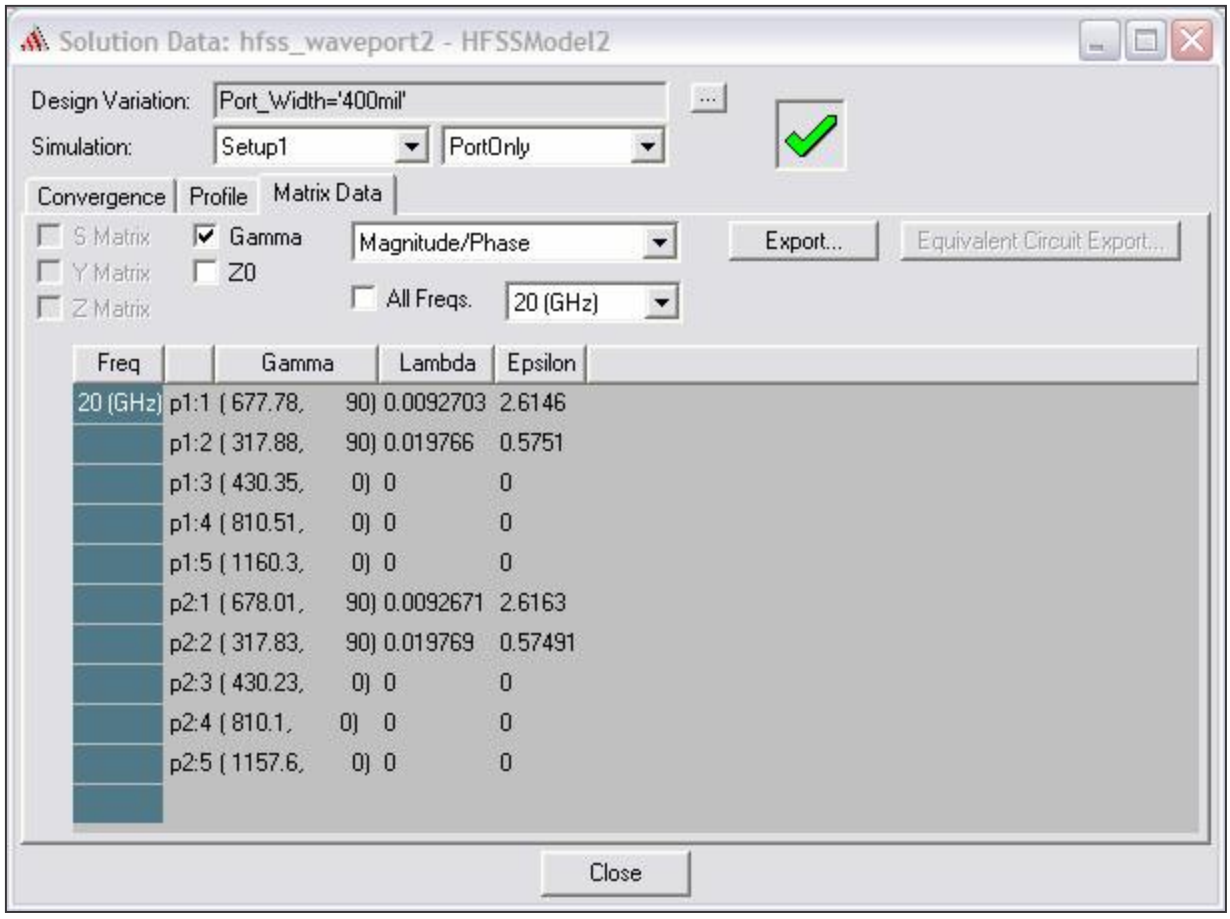
Analyze

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



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 Matrix Data:
       1. Click the **Matrix Data** Tab
  2. Click the **Close** button

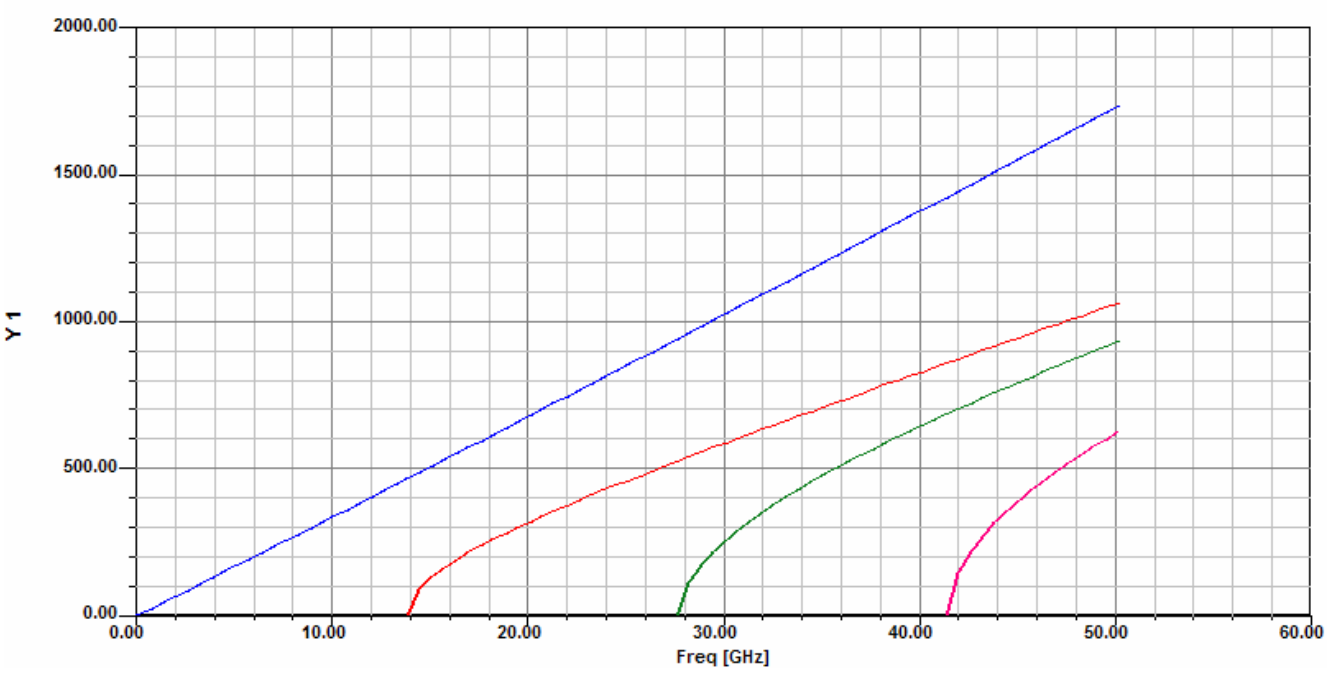


Create Reports

Create Propagation Constant vs. Frequency

- To Create a report:
  1. Select the menu item *HFSS > Results > Create Report*
  2. Create Report Window:
     1. Report Type: **Modal S Parameters**
     2. Display Type: **Rectangular**
     3. Click the **OK** button
  3. Traces Window:
     1. Solution: **Setup1: Sweep1**
     2. Click the **Y** tab
        1. Domain: **Sweep**
        2. Category: **Gamma**
        3. Quantity: **Gamma(p1:1),Gamma(p1:2), Gamma(p1:3), Gamma(p1:4), Gamma(p1:5)**
        4. Function: **im**
        5. Click the **Add Trace** button
     3. Click the **Done** button
     4. See next page for plot and discussion of results

Create Reports

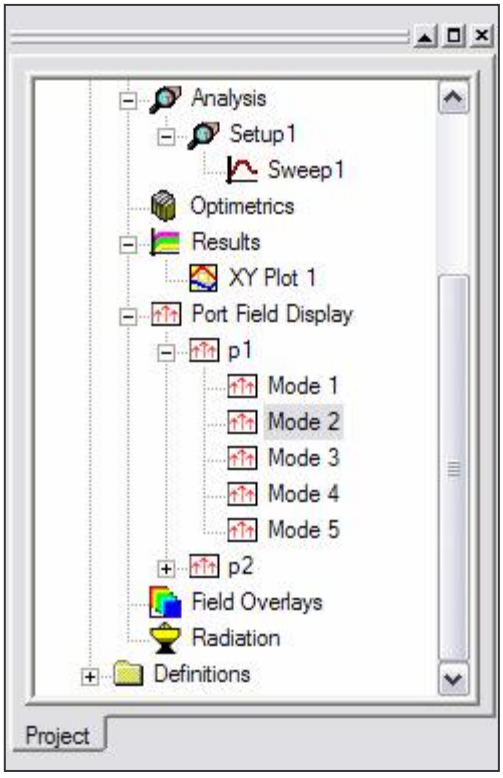


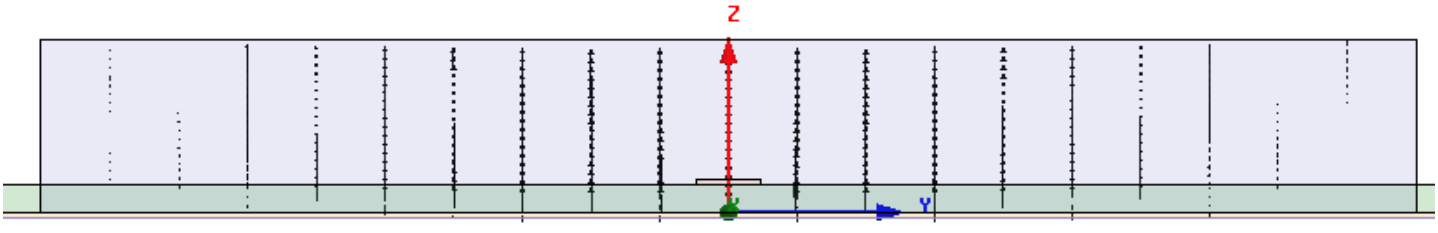
Discussion

- What does this plot tell us?
- Given the physical size of the wave port object that we created, the fundamental mode (p2:1) is a quasi-TEM mode that propagates from DC on up.
- This also shows that higher order modes can propagate (b>0) at a high enough frequency. The second modes starts propagating at ~14 GHz.
- Therefore, if we only needed to simulate up to 10 GHz, we wouldn't need to size our port any different, however, if we need to simulate up to 50 GHz, then we need to resize our port to eliminate the higher order propagating modes.
- The last part of the exercise will explore this.

Field Pattern Plots

- It might be illuminating to look at the field patterns at the port to discover which modes are excited.
- **To display the Mode field patterns in the modeler:**
  1. From within the project manager, expand the section **Port Field Display**
  2. Expand **p1**, and select **Mode 2**
     1. The field pattern will be overlaid on the 3D model. From the plot below, Mode 2 is clearly not a microstrip mode, but a TE_{10}-like waveguide mode.





Optimetrics Setup - 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. Variable: **Port_Width**
            2. Select **Linear Step**
            3. Start: **200mil**
            4. Stop: **600mil**
            5. Step: **100mil**
            6. Click the **Add >>** button
            7. Click the **OK** button
    2. Click the **OK** button

Save Project

- To save the project:
  1. In an Ansoft HFSS window, select the menu item *File > Save*.

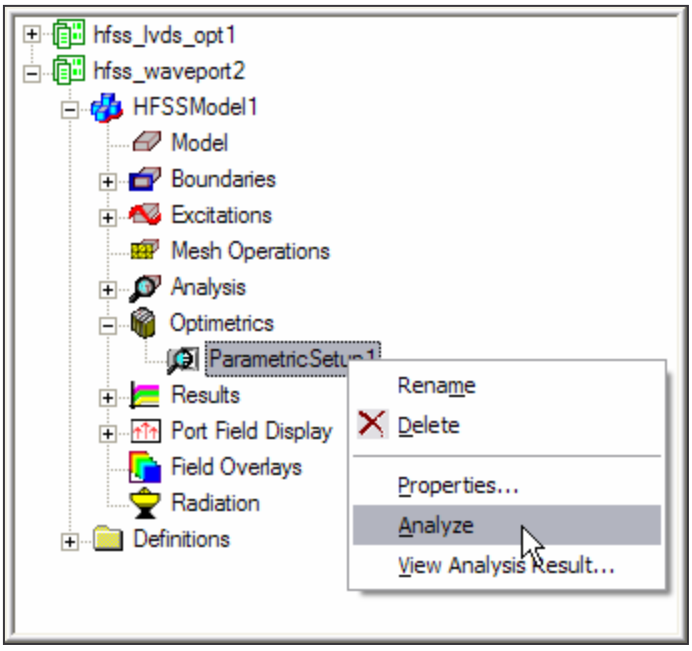
Analyze

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. Expand the Project Tree to display the items listed under **Optimetrics**
  2. Right-click the mouse on **ParametricSetup1** and choose **Analyze**

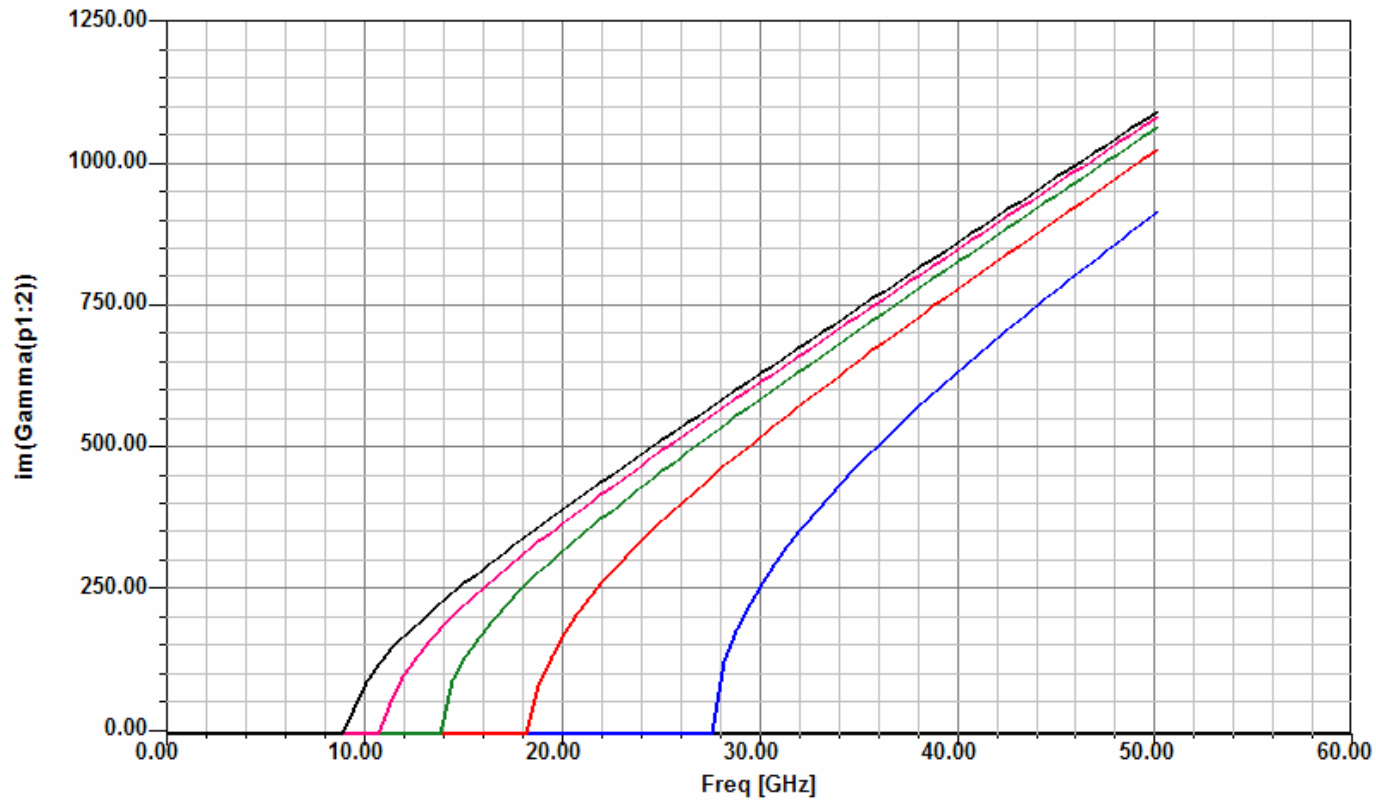


Create Reports

Create Propagation Constant vs. Frequency vs. Port_Width

- To Create a report:
  1. Select the menu item *HFSS > Results > Create Report*
  2. Create Report Window:
     1. Report Type: **Modal S Parameters**
     2. Display Type: **Rectangular**
     3. Click the **OK** button
  3. Traces Window:
     1. Solution: **Setup1: Sweep1**
     2. Click the Sweeps tab
        1. Select the **Sweep Design and Project Variable Values** button
     3. Click the Y tab
        1. Domain: **Sweep**
        2. Category: **Gamma**
        3. Quantity: **Gamma(p1:2),**
        4. Function: **im**
        5. Click the **Add Trace** button
     4. Click the **Done** button
     5. See next page for plot and discussion of results

Create Reports



Discussion

- What does this plot tell us?
- This shows that as we decrease the width of the port, the frequency at which the higher order mode starts to propagate increases.
- Therefore, by properly sizing the wave port, you can eliminate any higher order propagating modes if you believe that they do not exist.
- You need to use caution if you are simulating very high frequencies, i.e., millimeter wavelengths, as you may not be able to make the ports small enough to eliminate these modes. You probably shouldn't even try as the higher order modes might represent real world effects.