Getting Started with HFSS: Floquet Ports

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

Release 17.2
July 2016

ANSYS Inc. is
certified to ISO
9001:2008.

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.

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 online help from the 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.

Context-Sensitive Help

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

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

Table of Contents

1 - Introduction

This document is intended as supplementary material to HFSS users. We assume you have some experience of designing projects using HFSS. This tutorial includes instructions to create, solve, and analyze results of models that use a Floquet port.

This chapter contains the following topics:

- ✔ General Outline
- ✔ Floquet Ports in HFSS

General Outline

This tutorial teaches you how to perform the following tasks in HFSS:

- Create the geometric models.
- Modify the models' design parameters.
- Specify solution setup and sweep for the designs.
- Validate the design setups.
- Run HFSS simulations.
- Create 2D x-y plots of S-parameter results.

Floquet Ports in HFSS

The Floquet port in HFSS is used exclusively with planar-periodic structures. Chief examples are planar phased arrays and frequency selective surfaces when these may be idealized as infinitely large. The analysis of the infinite structure is then accomplished by analyzing a unit cell. Linked boundaries most often form the side walls of a unit cell, but in addition, a boundary condition is required to account for the infinite space above. The Floquet port is designed for this purpose.

The Floquet port is closely related to a Wave port in that a set of modes ("Floquet modes") represents the fields on the port boundary. Fundamentally, Floquet modes are plane waves with propagation direction set by the frequency, phasing, and geometry of the periodic structure. Just like Wave modes, Floquet modes too have propagation constants and experience cut-off at low frequency.

When a Floquet port is present, the HFSS solution includes a modal decomposition that gives additional information on the performance of the radiating structure. As in the case of a Wave port, this information is cast in the form of an S-matrix interrelating the Floquet modes. In fact, if Floquet ports and Wave ports are simultaneously present, the S-matrix will interrelate all Wave modes and all Floquet modes in the project.

For the current version, the following restrictions apply:

- Currently, only modal projects may contain Floquet ports.
- Boundaries that are adjacent to a Floquet port must be linked boundaries.
- Fast frequency sweep is not supported. (Discrete and interpolating sweep are supported.)

2 - Array Antennas

This chapter contains the following topics:

Sample Project - Array Antenna

"Array Antenna" on the next page. illustrates an infinite array of square aperture antennas in a conducting plane. The elements are fed from below the plane by square waveguide ports with dominant modal field aligned in the direction shown.

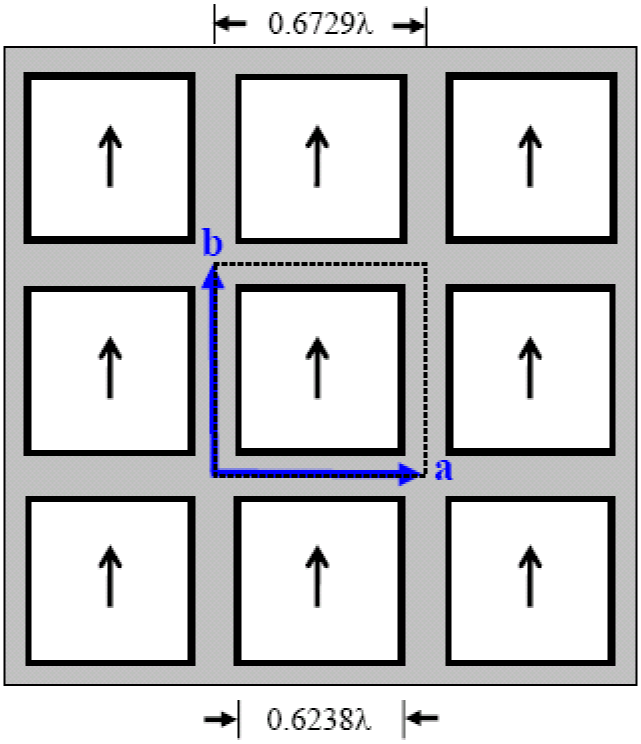


Figure 2-1 Array Antenna

When the elements are uniformly excited so that the array radiates in the broadside direction, the field above the array is periodic. The relative positions of the periodic points where field values are equal may be specified by a pair of vectors, shown in blue and denoted by a and b. These “lattice vectors" describe the geometry of the array but are independent of the nature of the array elements themselves. The lattice vectors may be parallel-transported anywhere in the array plane to show corresponding field points.

In the usual way, a unit cell may be analyzed to represent the entire array and the outline of one possible unit cell is shown in "Array Antenna" above. using dotted lines.

"HFSS model of unit cell with lattice vectors " on the facing page. depicts an HFSS model for the unit cell of the infinite array. The model consists of two boxes. The lower box represents the feeding waveguide and the top box is the unit cell for the region above the plane. The dimensions and geometry of the unit cell reflect the lattice vectors of the array. Linked boundaries are defined on the cell walls and a Wave port provides the array element excitation.

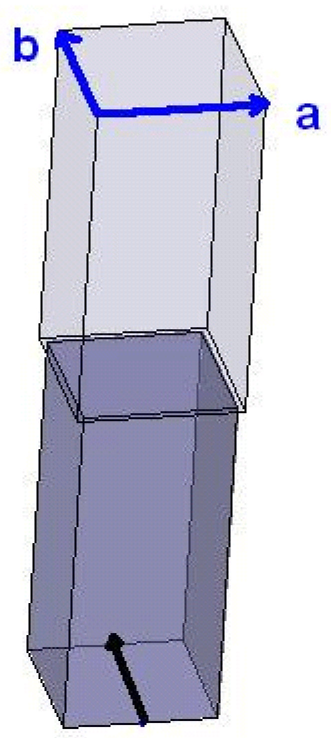


Figure 2-2 HFSS model of unit cell with lattice vectors

Note A Floquet port is used on the (top) open boundary.

Set Units and Solution Type

Set the units and solution type for this project before you design the model.

1. Open a new Project and name it *rhombicArray*.
2. Click **Modeler**>**Units** and set the units to *cm*and click **OK.**
3. Insert an HFSS design type.
4. Click **HFSS**>**Solution Type**.
5. Select **Modal**.
6. Click **OK**.

Create the Unit Cell Model for the Array Antenna

1. Click **Draw>Box** to create an arbitrary box and edit the fields in the **Properties** dialog box as in "Command tab of the Properties dialog box" on the next page. .

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-0.33645 ,-0.33645 ,0	meter	-0.33645meter , -0.33645meter , 0meter
	XSize	0.6729	meter	0.6729meter
	YSize	0.6729	meter	0.6729meter
	ZSize	1.4	meter	1.4meter

Figure 2-3 Command tab of the Properties dialog box

2. Select the newly created box, and change **Transparency** to 0.86.
3. Click **Draw>Box** to create a second arbitrary box, and edit the fields as in "Command tab for the second box." below. .

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-0.3119 ,-0.3119 ,0	meter	-0.3119meter , -0.3119meter , 0meter
	XSize	0.6238	meter	0.6238meter
	YSize	0.6238	meter	0.6238meter
	ZSize	-1.4	meter	-1.4meter

Figure 2-4 Command tab for the second box.

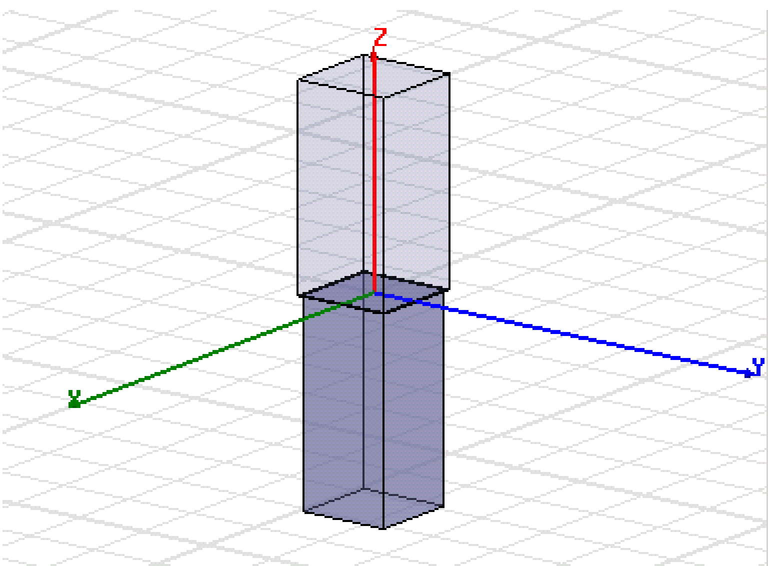


Figure 2-5 The Unit Cell Model

Assign Master and Slave Boundaries

Assign boundaries to the box object as follows.

1. Hit **F** to enter face selection mode.
2. Select the face (see "U Vector drawn" on the next page.) of the first box, and click **HFSS>Boundaries>Assign>Master.**

   The **Master Boundary** dialog box appears.

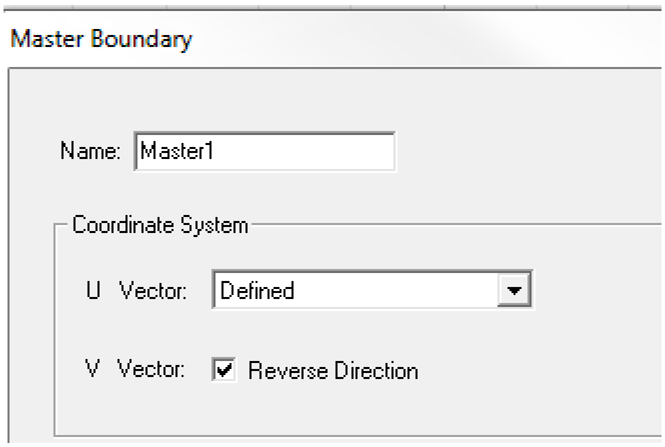


Figure 2-6 Master Boundary dialog

3. Select *New Vector* from the **U-Vector** drop-down menu.

   Note Leave the default **Name** as *Master1*.

   The **Measure Data** and **Create Line** dialog boxes appear.

   Note Measure Data will show up only if you have checked “**Show measure dialog**” on the **Drawing** tab of the **Modeler Options** (under **Tools**) dialog box.

4. Click the lower right-most corner of the face and draw the U vector (red arrow) along the left. See "U Vector drawn" on the next page. .

   Note Draw the vector from right to left.

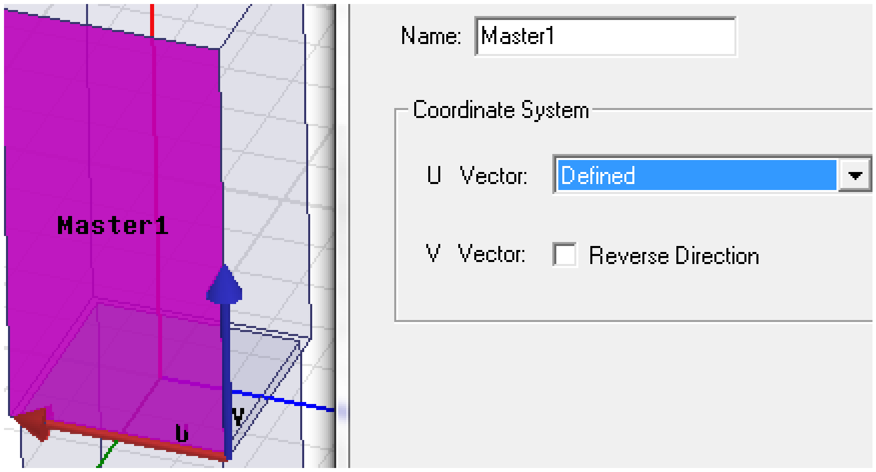


Figure 2-7 U Vector drawn

5. Click **OK**.

   The *Master 1* boundary gets assigned. See "Boundary applied" below. .

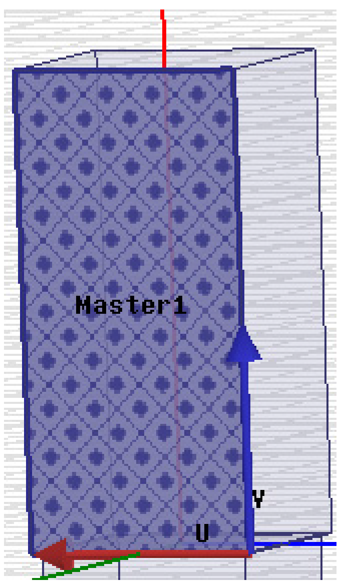


Figure 2-8 Boundary applied

6. Press **Alt** and rotate the box 180 degrees to access the face opposite to *Master1*. See "Face with the prospective Slave1 boundary" on the facing page. .

   Note You can also select another face, and hit **B** to access the desired face.

7. Click **HFSS > Boundaries > Assign > Slave**.

   The **Slave: General Data** dialog box appears.

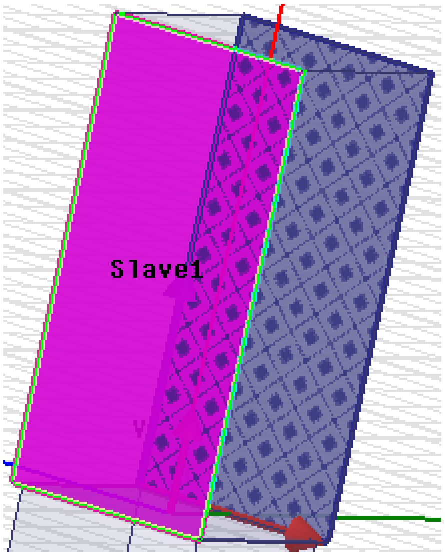


Figure 2-9 Face with the prospective Slave1 boundary

8. Select *Master 1* on the **Master Boundary** drop-down menu.
9. Select *New Vector* on the **U Vector** drop-down menu.

   Note Leave **Name** as *Slave1*.

   The **Measure Data** and the **Create Line** dialog boxes appear with the cursor holding the dotted line.
10. Click the left-most corner of the face and drag the cursor to the right-most corner, click, again.

    Note Remember the direction of the U-V vectors for the Master must be the same for its corresponding Slave.

    The **Slave: General Data** dialog appears with the U and V vectors applied.
11. Verify that the option **Reverse Direction** of V-Vector is checked.

    Note The **U-Vector** field should show as *Defined* now.
12. Click **Next**.

    The **Slave: Phase Delay** dialog box appears.
13. Click **Finish**.

    The *Slave1* boundary is assigned.

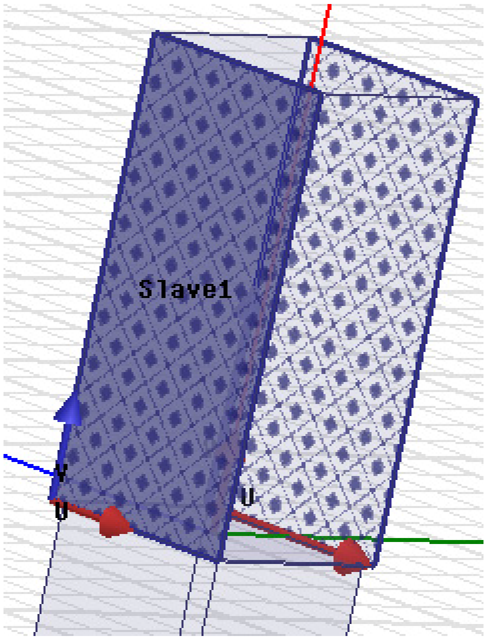


Figure 2-10 The Slave1 boundary is assigned.

14. Right click the face on the right side of *Slave1* and select **Assign Boundary> Master** from the short-cut menu.

    The **Master Boundary** dialog box appears.
15. Select *New Vector* on the **U-Vector** drop-down menu.
16. Click the leftmost corner of the face and drag the cursor to the rightmost corner to draw the U-Vector.

    The U-vector arrow is applied whereas the V-vector arrow points downwards. See "The V-Vector pointing downwards." on the facing page. and "Master Boundary" on the facing page. .
17. Check **V-Vector Reverse Direction** as in "Master Boundary" on the facing page. .

    Note You need to reverse the direction of the V-Vector to include it on the box and not outside of it. For more information, see the next section.

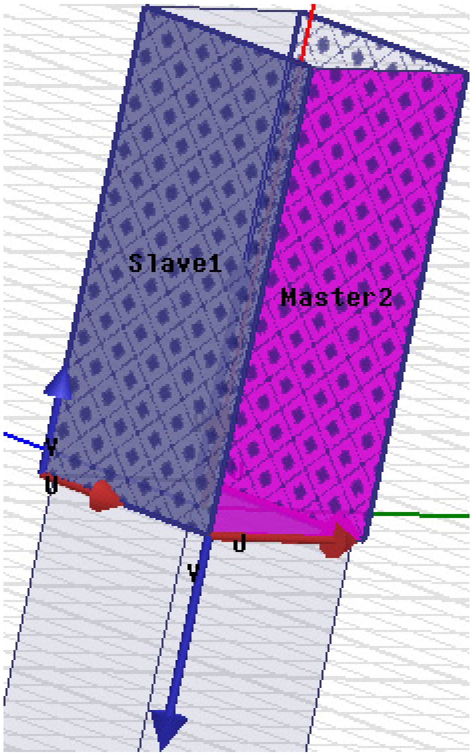


Figure 2-11 The V-Vector pointing downwards.

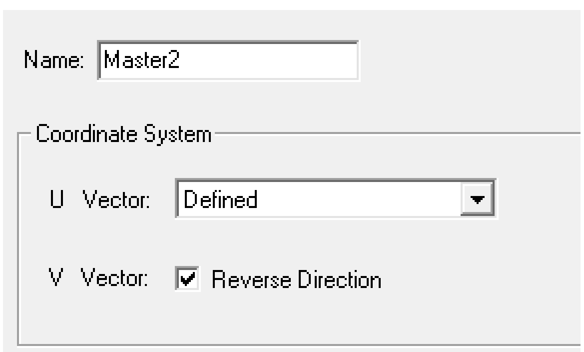


Figure 2-12 Master Boundary

18. Check **V Vector:Reverse Direction**.
19. Click **OK**.

    The direction of the V-Vector is reversed and the boundary *Master2* is applied.
20. Rotate the box 180 degrees to access the face opposite to *Master 2*.
21. Right click> A**ssign Boundary >Slave.**

    The **Slave: General Data** dialog box appears.
22. Select *Master2* as the **Boundary** and *New Vector* as the **U Vector**.

The **Create Line** and **Measure Data** boxes occur.

23. Click the rightmost corner of the face and drag the cursor to the left most corner, click and then, release.

    The **Slave: General Data** dialog box appears.

24. Click **Next.**

    The **Slave: Phase Delay** dialog box appears.

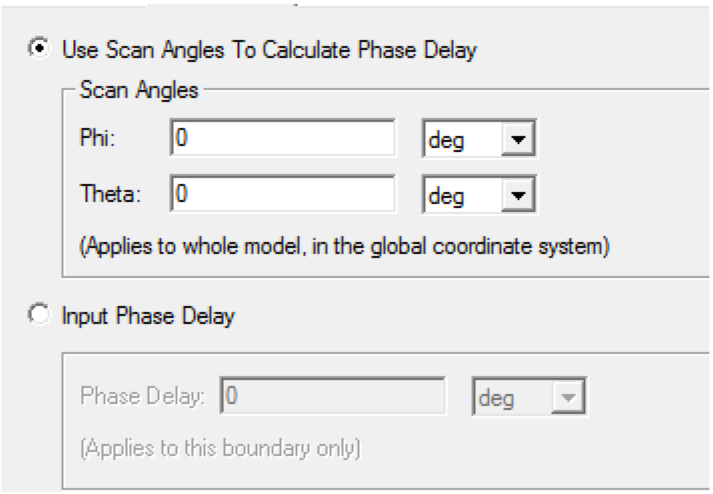


Figure 2-13 Phase Delay

25. Accept the default settings and click **Finish**.

    The *Slave2* boundary is assigned.

Direction of the U-V Vectors

HFSS uses the U-V vectors to set up a local co-ordinate system. A point on the Master boundary must correspond to that on the Slave. A point on the Slave needs to be paired on the Master so that a one-to-one correspondence can be established. Such a co-ordinate system should be constructed and aligned properly in the same direction so that the mapping is accomplished successfully. Therefore, when the V-vector is directed downwards, you must check the "**Reverse Direction**" option to keep it on the face of the box and not pointing outside of it.

Assigning Wave Ports

HFSS offers you both Integration lines and a localized co-ordinate system to assign Wave ports. When assigning a wave port HFSS generates a **Wave Port: Modes** dialog box with options to help set-up the **Mode Alignment and Polarity**. A portion of this dialog box is shown in "Portion of the Wave Port: Mode dialog box" on the facing page. .

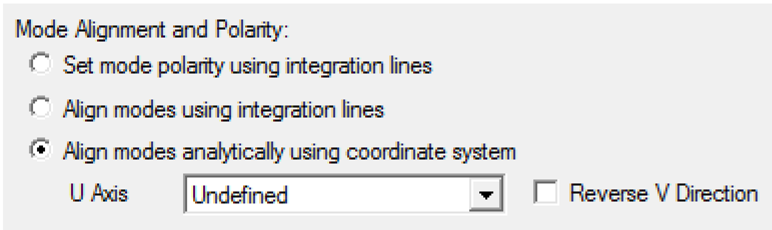


Figure 2-14 Portion of the Wave Port: Mode dialog box

Note This dialog box will appear only *while* you assign Wave ports.

For this particular design i.e. the cell model with emphasis on Floquet ports, we recommend you assign the Wave port with analytical mode alignment. The last radio button in "Portion of the Wave Port: Mode dialog box" above. should be selected for this project.

Note For more information, see the online help.

Assign Wave Port with Analytical Mode Alignment

A Wave port boundary will be assigned to the bottom surface of the feeding waveguide to model the energy injected into the array element.

1. Select the bottom face of the lower box. See "Wave Ports assigned on the bottom face" on page 2-13 . .
2. Right-click, and from the shortcut menu, select **Assign Excitations > Wave Port**.

   The **Wave Port: General** dialog box appears.
3. Enter a*wave port* in the **Name** field and click **Next**.

   The **Wave Port: Modes** dialog box appears.
4. Make the following entries:
   - Enter **Number of Modes** as *2*.

   Note Because the feeding waveguide has a square cross section, two waveguide modes must be specified.

   - Select the radio button “**Align Modes analytically using co-ordinate system.**”
   - Select “*New Vector*” from the **U Axis** drop-down menu.

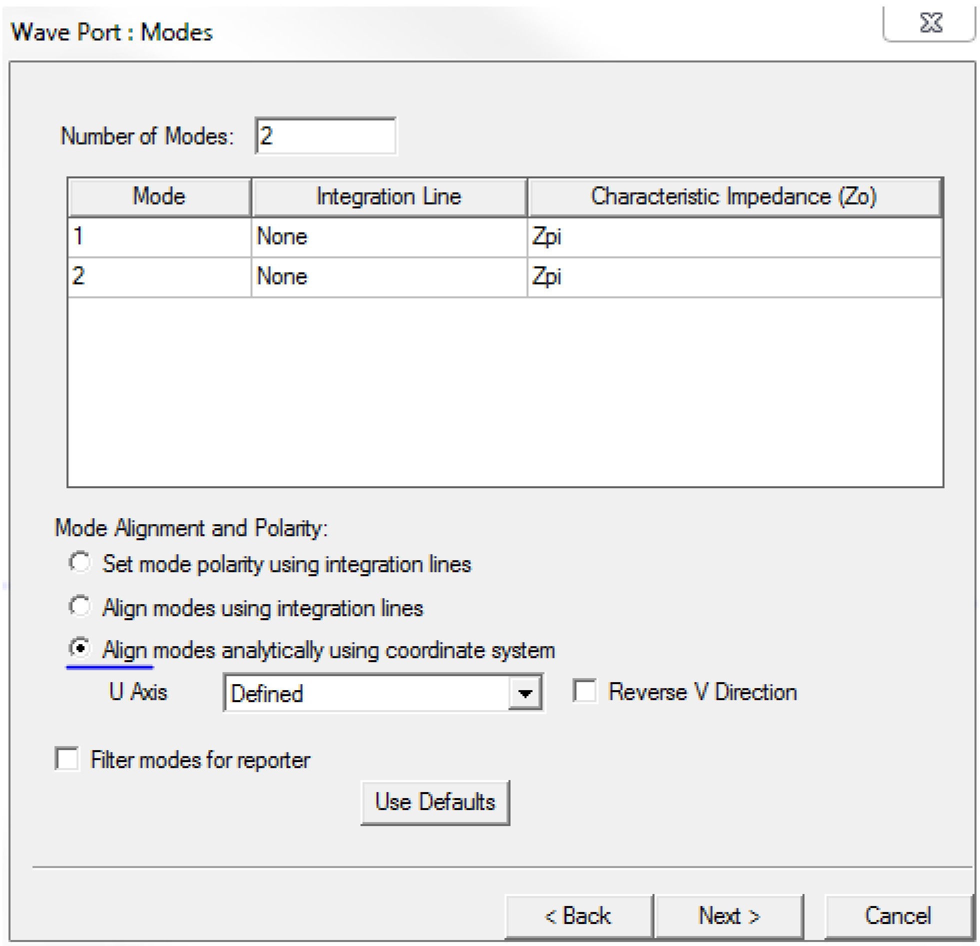


Figure 2-15 Wave Port: Modes Dialog Box - see *blue-ink that highlights the radio button*.

The cursor with a dotted line appears.

5. Click the mid point of a side (nearest to the X axis) of the bottom face.

   A small opaque triangle and mini axes appear.

6. Draw the line by following the horizontal mini axis (perpendicular to the X axis and therefore, parallel to the Y axis. See "Wave Ports assigned on the bottom face" on the facing page. .

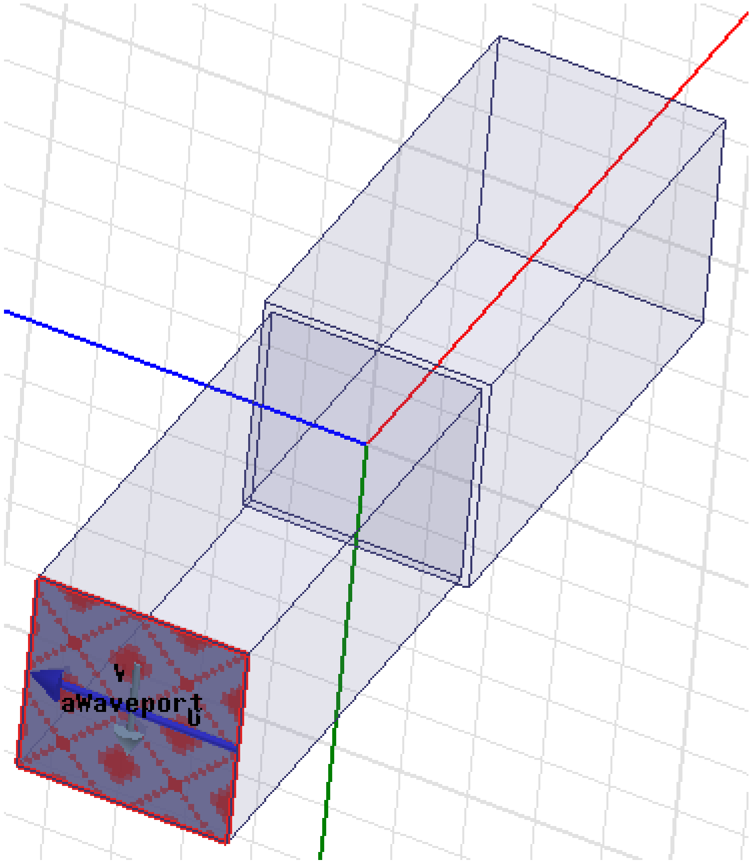


Figure 2-16 Wave Ports assigned on the bottom face

The **Wave Port: Modes** dialog box appears.

7. Select "**Reverse V Direction**."

   V vector points in the direction of the positive Y-axis.

8. Click **Next** on the **Wave Port: Modes** dialog box.

   The **Wave Port: Post Processing** dialog box appears.

9. Verify that the default settings are as shown in "Wave Port: Post Processing" on the next page. .

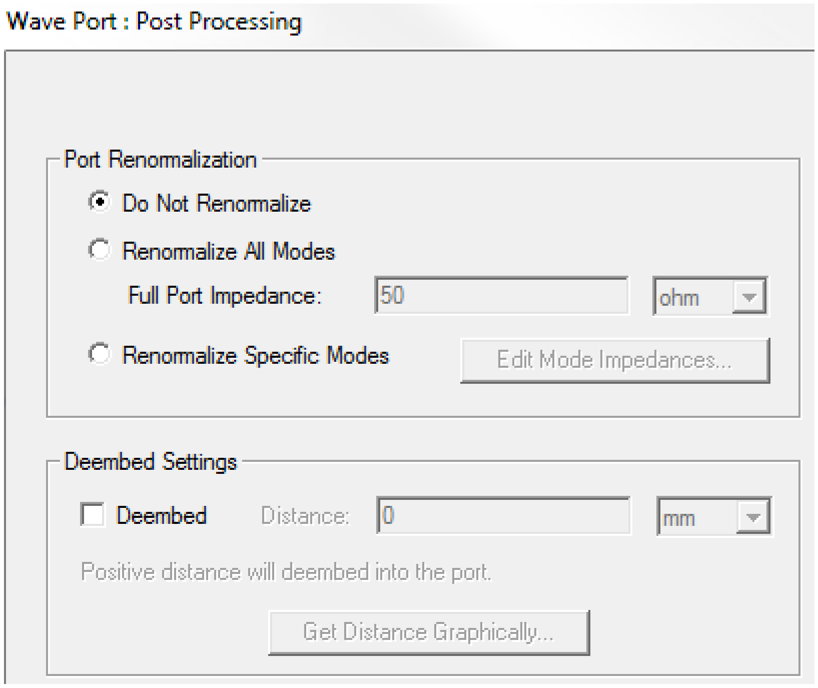


Figure 2-17 Wave Port: Post Processing

10. Click **Finish**.

    The excitation is assigned and **a wave port** occurs under **Excitations** on the **Project Manager** window.

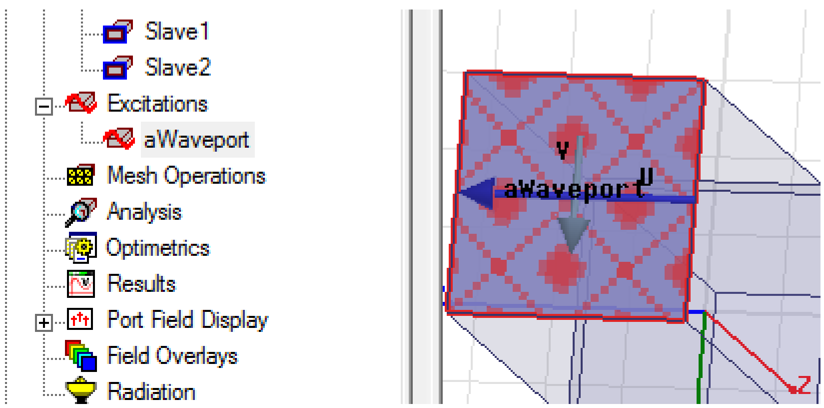


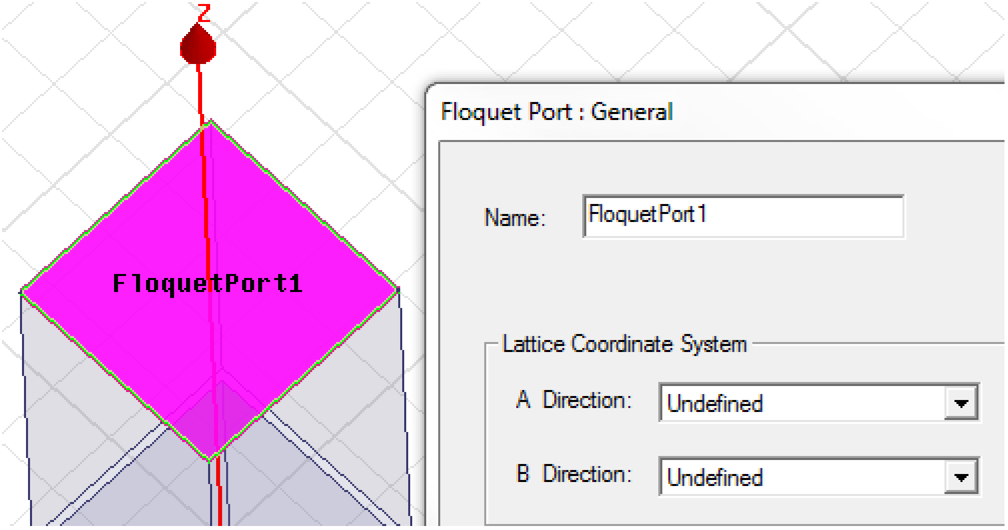
Figure 2-18 Wave Ports assigned

Assign Floquet Port

To assign the Floquet port perform the following steps.

1. Select the top face of the upper box.
2. Right click, and select **Assign > Excitation > Floquet Port** from the shortcut menu.

   The **Floquet Port: General** dialog box appears and the top face gets labeled as **Flo-**



   quetPort1.

Figure 2-19 Floquet Port: General dialog box

3. On the **Lattice Coordinate System**, select *New Vector* for the **A Direction**.

   The cursor with the new dotted line appears on the Modeler window.
4. Draw the **A-Direction** vector parallel to the X-axis with the arrow in the same direction that of the positive x-axis.

   The "**a"** vector is drawn and the **Floquet Port: General** dialog appears.

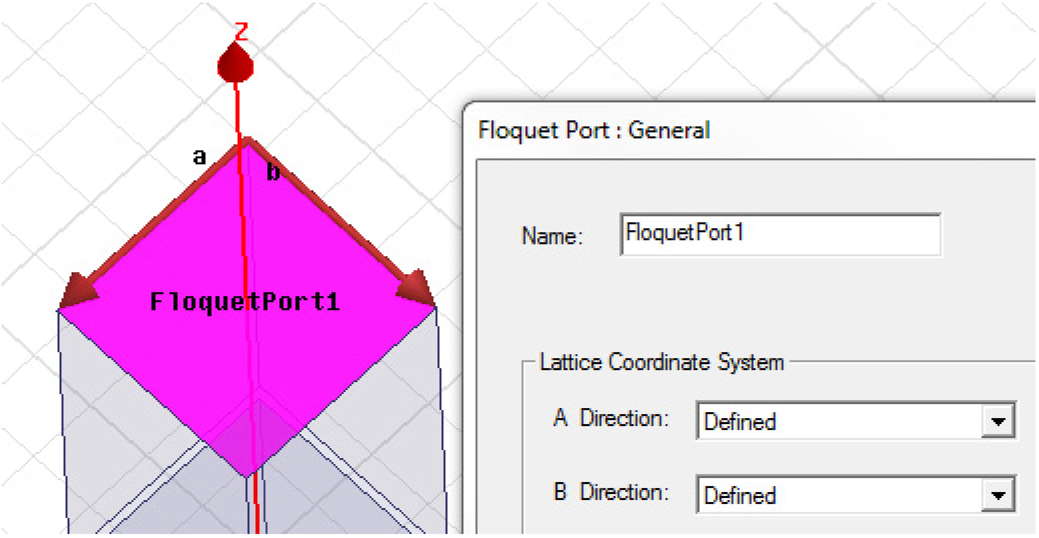


Figure 2-20 Floquet ports assigned

5. Select *New Vector* from the **B-Direction** drop-down.
6. Draw the B vector perpendicular to the A vector.

   The **Floquet Port: General** dialog box showing both the vectors as "*Defined*" recurs.

   Note The lattice vectors **a** and **b** must have a common origin.

7. Click **Next**.

   The **Floquet Port: Modes Setup** dialog box appears.

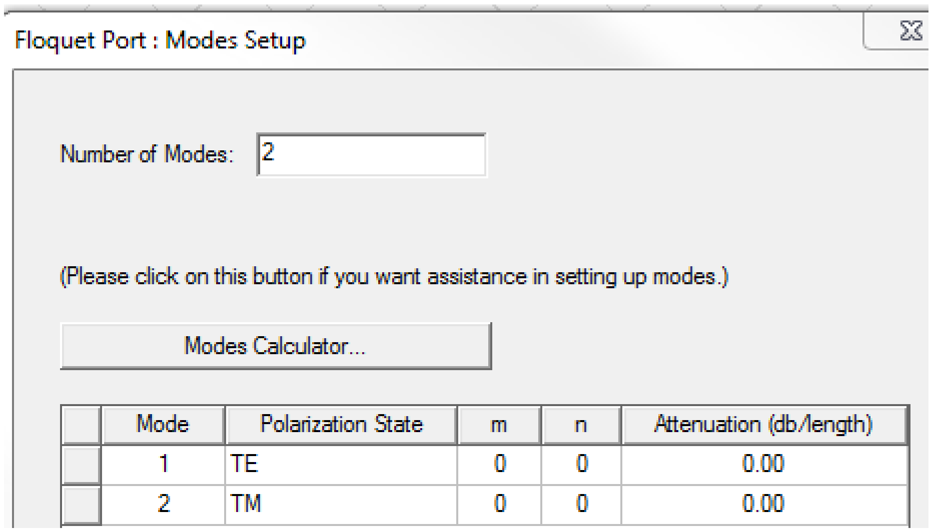


Figure 2-21 Modes Set up

8. Click **Next**.

   The Post Processing dialog box appears.

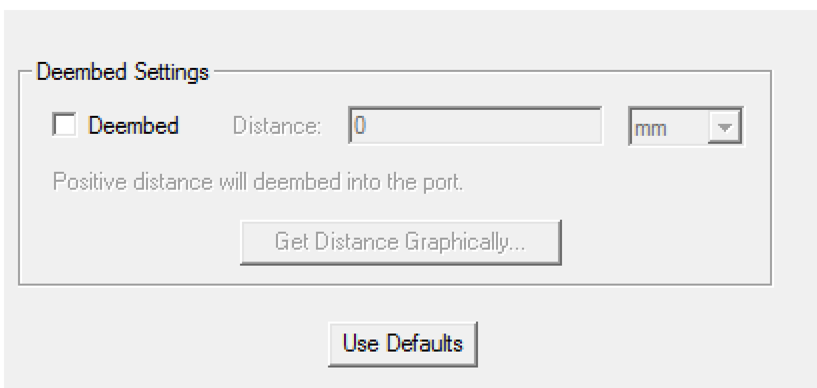


Figure 2-22 M>Post Processing

9. Verify that the Deembed box is unchecked and click **Next**.

   The 3 D Refinement dialog box appears.

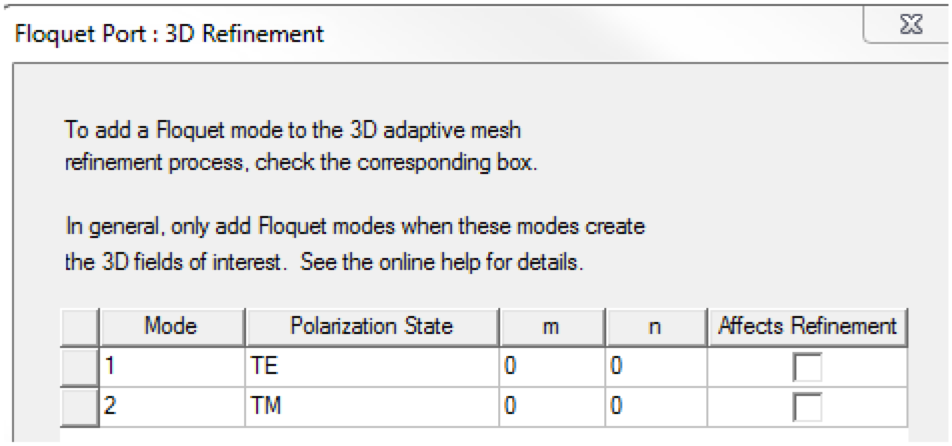


Figure 2-23 Floquet Port: 3-D Refinement

10. Click **Finish**.

    The Floquet port is assigned and **FloquetPort1** appears in the Project Manager window under **Excitations**.

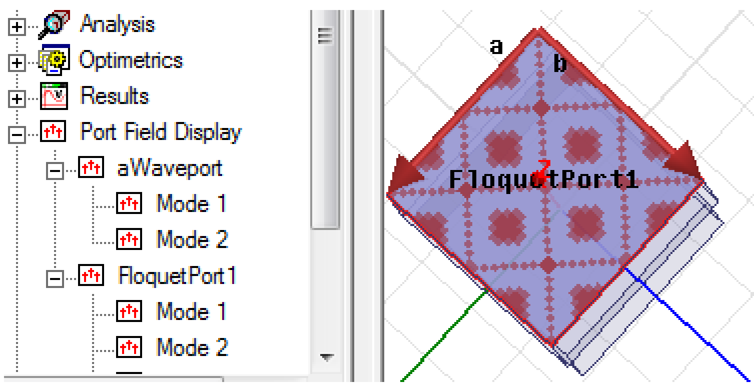


Figure 2-24 Figure 24. Floquet Port Assigned

Floquet Port Dialog Box

While assigning the Floquet Ports, you used the different options available on the Floquet Port dialog box. The following sections describe the settings on each of these tabs.

Floquet Port: Mode Setup

See "Modes Set up" on the previous page. Floquet modes are specified by two modal indices and a polarization setting. These designations resemble the textbook notation for rectangular waveguide modes, such as "TE_{10}". The Floquet field patterns are different, however.

The Floquet modes in the modes table both have modal indices m and n equal to zero. They are sometimes called “specular” modes. In this project, the specular modes consist of two orthogonally-polarized plane waves propagating normal to the plane of the array.

Specular modes are always an essential part of the Floquet mode set. However, sometimes due to electrical symmetry, one of the two polarizations may be omitted.

The final column of the mode table is labeled “Attenuation”. The attenuation of the mode is measured in the direction normal to the plane of the array.

The value given for the specular modes is 0 dB since they are propagating. Higher-order modes will experience attenuation if they are cut off. The value of attenuation is greater than zero.

Note The default specular pair of modes is sufficient for this simulation.

Floquet Port: Post Processing

See "M>Post Processing" on page 2-16 . . This tab lets you specify a de-embedding distance. Just as in a Wave port, de-embedding is an optional post processing step employed when the phase of the S-parameter elements is of interest. The interface for de-embedding a Floquet port is the same as that for a Wave port. For this example, we omit specifying any de-embedding.

Floquet Port: 3D Refinement

See "Floquet Port: 3-D Refinement" on the previous page. . The 3D Refinement tab controls which Floquet modes participate in 3D adaptive mesh generation.

The default settings are such that Floquet modes will not participate in 3D adaptive refinement.

In any model with Wave, Lumped, or Floquet ports, there is a 3D modal field pattern associated with each defined mode. This is the 3D field pattern corresponding to stimulation of the one mode, with all other modes match terminated. The 3D mesh that is created by HFSS on subsequent adaptive passes is actually a compromise designed to adequately represent all modal field patterns simultaneously.

When a specific modal field pattern is of interest, greater solution efficiency is achieved by adapting only on the basis of the one field pattern. In the case at hand, the modal pattern corresponding to excitation of the Wave port is of prime interest. Hence, the default setting of the 3D refinement tab which ignores the Floquet modal field patterns in adapting the 3D mesh.

Add Solution Setup

You must add a Solution Setup for your model.

1. Right click **Analysis** in the Project tree, and select **Add Solution Setup**.

   The **Solution Setup** dialog box appears.

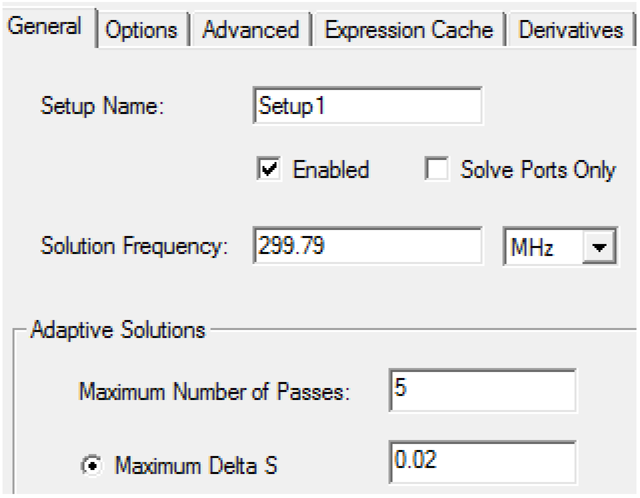


Figure 2-25 Solution Setup: General

2. Enter the settings as shown in "Solution Setup: General" above. .
3. Click **Options**.
4. Enter the settings as in following figure.

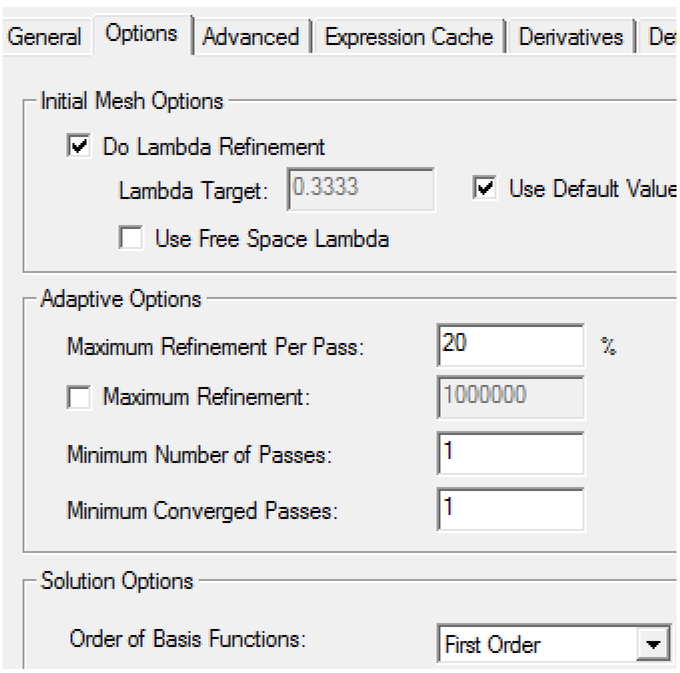


Figure 2-26 Solution Setup: Options

5. Click **OK** to accept the settings.

Run Simulation and View Results

Now that you have completed the entire set-up you are ready to analyze it. Before that, be sure to save the project.

1. Click **HFSS>AnalyzeAll**.

   The Progress Window indicates HFSS is running the simulations. The Message Manager indicates normal completion of the simulation if everything went off well.

2. Right click **Results** icon in the Project tree, and select **Solution Data**.

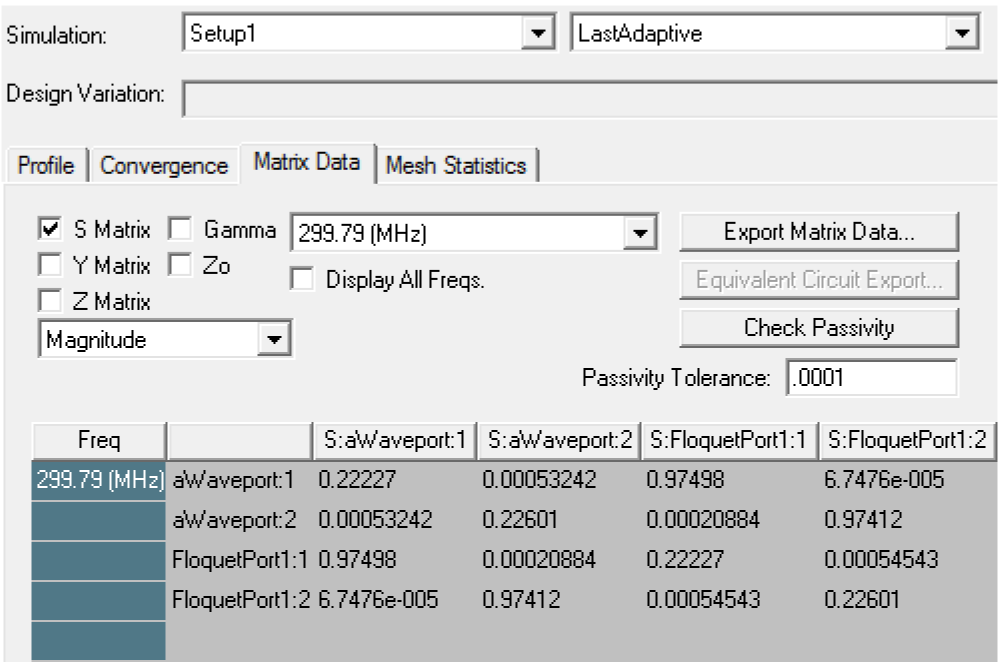


Figure 2-27 HFSS Design Solutions

"HFSS Design Solutions" above. shows the post-simulation matrix data panel. Since phase is unimportant for this case, only magnitudes are shown. You can select *Magnitude* from the drop-down menu under **Z Matrix** check box. There are several things to note.

- The S-matrix is a 4×4 matrix and interrelates the wave port modes with the Floquet modes.
- The Floquet modes in the S-matrix are listed next in the order specified in the Floquet port setup panel. By referring to this panel, we are therefore reminded that FloquetPort1:1 refers to the TE00 Floquet mode and FloquetPort1:2 refers to the TM00 Floquet mode.
- The first column of the matrix gives the modal amplitudes for a unit stimulation of the Wave port, i.e. the array acting in “transmit mode”. The first entry, 0.22227, is the magnitude of the active reflection coefficient of the unit cell. The third and fourth (row) entries give the power coupled into each of the two Floquet modes. Virtually all the non-reflected power is coupled into the first Floquet mode. The excited Floquet mode represents a plane wave leaving the face of the array.

- The third and fourth columns of the S-matrix indicate the power couplings for individual excitation of each of the Floquet modes. The physical picture of an exciting Floquet mode is a plane wave incident on the infinite array from above. In the case at hand, each Floquet mode transfers most of its power to the wave port mode with the same polarization. In these cases think of the array acting in “receive mode”.

Create Variables for Scan Angles

Array antennas have the unique ability to redirect (scan) their energy without mechanical motion. To include the effects of scanning in the HFSS simulation, this section will discuss parameterizing the simulation using scan angles.

1. On the menu bar, click **Project>Project Variables**.

   The **Properties** dialog box appears.
2. Click **Project Variables** and then, the **Add**... button.

   The **Add Property** dialog box appears.

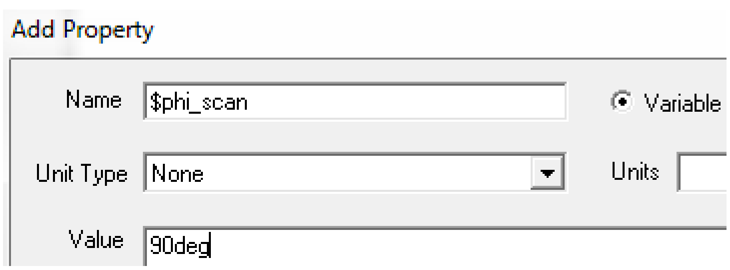


Figure 2-28 Add Property dialog box

3. Set **Name** to *$phi_scan*, and **Value** = 9*0 deg*.
4. Click **OK**.

   The **Add Property** dialog box closes and the variable *$phi_scan* gets added on the **Project Variables** window.
5. Click the **Add** button.

   The **Add Property** dialog box appears again.
6. Set **Name** to *$theta_scan* and **Value** = *0 deg*.
7. Click **OK**.

   The **Add Property** dialog box closes and the variable *$phi_scan* gets added on the **Project Variables** dialog box.

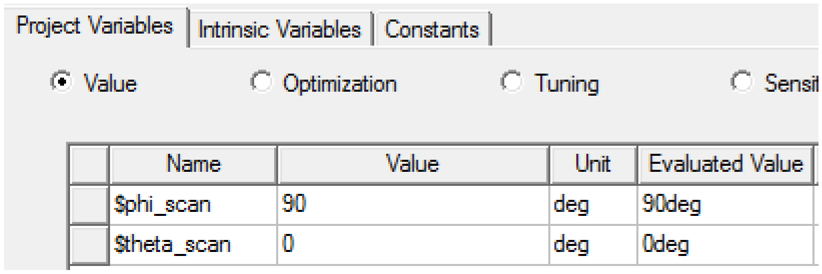


Figure 2-29 Project Variables dialog box

8. Click **OK** to close the project **Properties** window.

Note When using scan angles in unit cell models, the plane of periodicity (here the array plane) should be parallel to the global coordinate x-y plane. Although not absolutely mandatory, adhering to this natural convention insures the simplest and clearest relationship between scan angles and the structure being modeled.

Use Scan Angles for the Model

Now that you created the scan angle variables, you will apply them to the model. To use them in the model, perform the following steps.

1. Double-click **Slave1** from the Project Tree.

   The **Slave: General Data** dialog box appears.

2. Click **Phase Delay** and set the fields as shown in following figure.

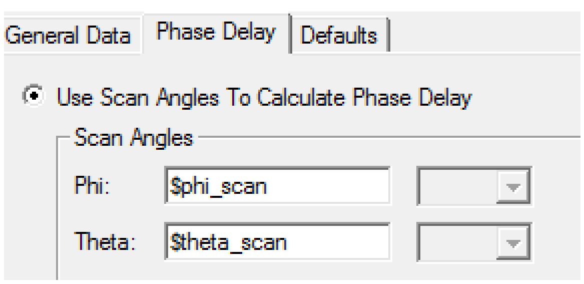


Figure 2-30 Use Scan Angles

3. Click **OK**.

   The scan angles are set and copied in the settings for Slave2.

Note Ignore the following informational message that will appear in the Message Manager window - “Solutions have been invalidated. Undo to recover.”

Parametric Sweep of Scan Angle

To illustrate more aspects of the Floquet port interface, we now seek the active reflection coefficient of the array as a function of scan angle. When the scan direction is in the E-plane of the array, the array exhibits a nice example of scan blindness at an angle of 27.5 degrees. To demonstrate this and other phenomena, we will set up a parametric sweep to make an E-plane scan from broadside (scan = 0) to endfire (scan = 90 degrees).

To setup a parametric analysis:

1. Right click **Optimetrics** and select **Add>Parametric**.

   The **Setup Sweep Analysis** dialog box appears.

2. Click **Add** on the **Sweep Definitions** tab.

   The **Add/Edit Sweep** dialog box appears.

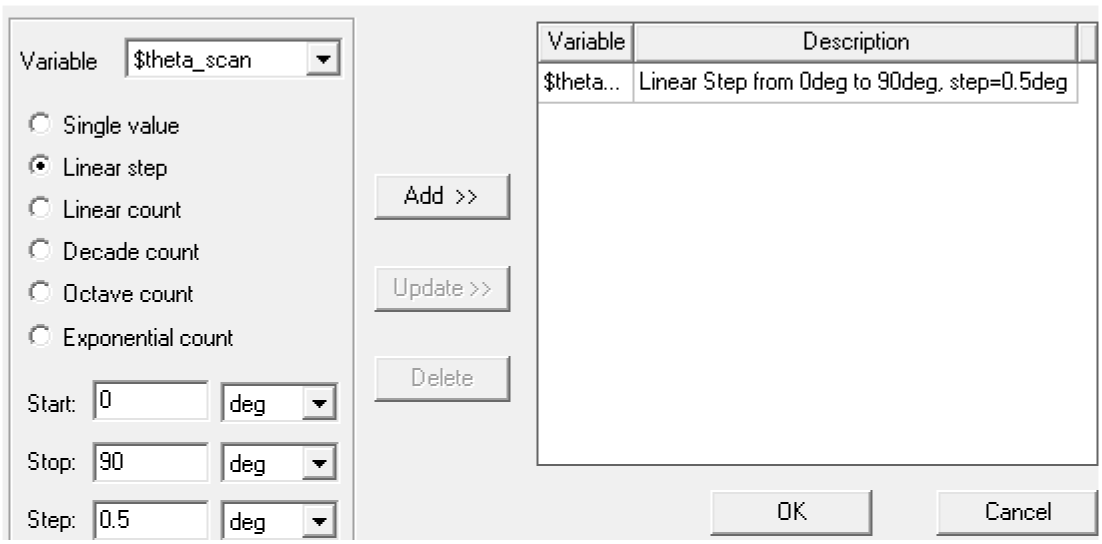


Figure 2-31 Add/Edit Sweep dialog box

3. Make the entries as in "Add/Edit Sweep dialog box" above. .
4. Click **Add >>** to include the settings.
5. Click **OK**.

   The **Setup Sweep Analysis** dialog box lists the *$theta_scan* variable and its description.

6. Click **General** to verify the settings are as in following figure.

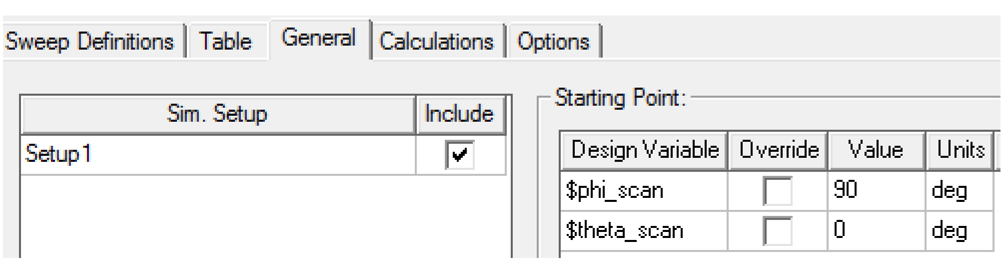


Figure 2-32 Setup Sweep Analysis

7. Click **Options.**
8. Set the fields as shown in following figure.

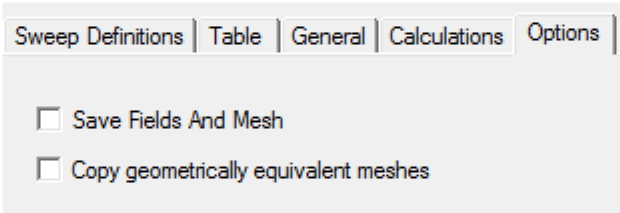


Figure 2-33 Options

9. Click **OK** to close the dialog box.

   Parametric Setup1 is listed under **Optimetrics**.

Set Up Modes for Parametric Sweep

1. Double-click **FloquetPort1** under the **Excitations**.

   The **Floquet Port** dialog box appears.
2. Click **Modes Setup**.

   Note A pair of modes is adequate for broadside scan. For larger scan angles, additional modes are required.
3. Click **Modes Calculator**.

   Note The inputs for mode selection do not affect the model set up and so the choices can be varied to explore different scenarios.
4. Enter 10 for **Number of modes** for the port. See "Mode Table Calculator" on the facing page. .

   Note Requesting so many modes will give you an idea what modes are necessary and what are not.
5. Set the **Frequency to 299.97 MHz**, which is the actual frequency of simulation.

   Note If the problem setup contained one or more frequency sweeps, usually this value is set to the highest frequency to detect higher-order propagating modes.
6. Enter **phi scan angle** as *90* degrees in both the **Start** and **Stop** fields.

   Note To create a set of modes that will be adequate for every scan direction in the parametric sweep, the scan angles of the sweep are entered in a start-stop-step format. The entered angles are spherical polar angles of the global coordinate system.
7. Enter the same values for the **Theta** fields with **Step Size** as **0.5**. See following figure.

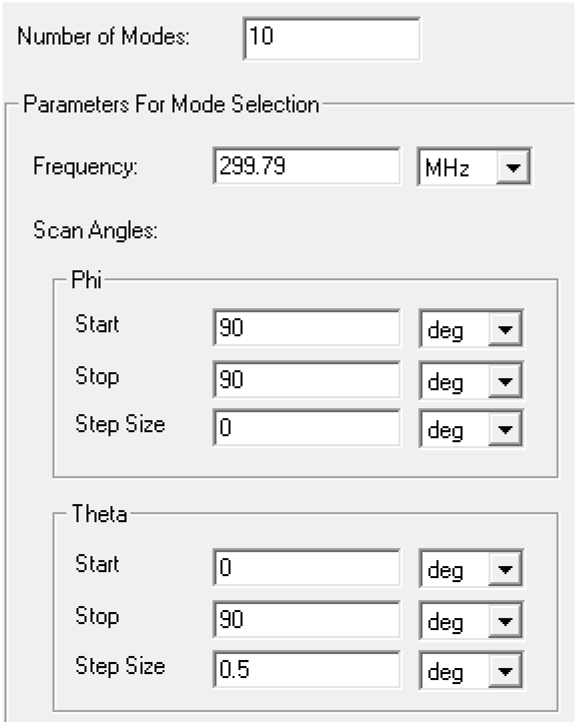


Figure 2-34 Mode Table Calculator

8. Click **OK**.

   The calculator closes and displays the results in the modes table. See "Floquet Port dialog open on Modes Setup" on the next page. .

9. Re-enter *4* in the **Number of Modes** field.

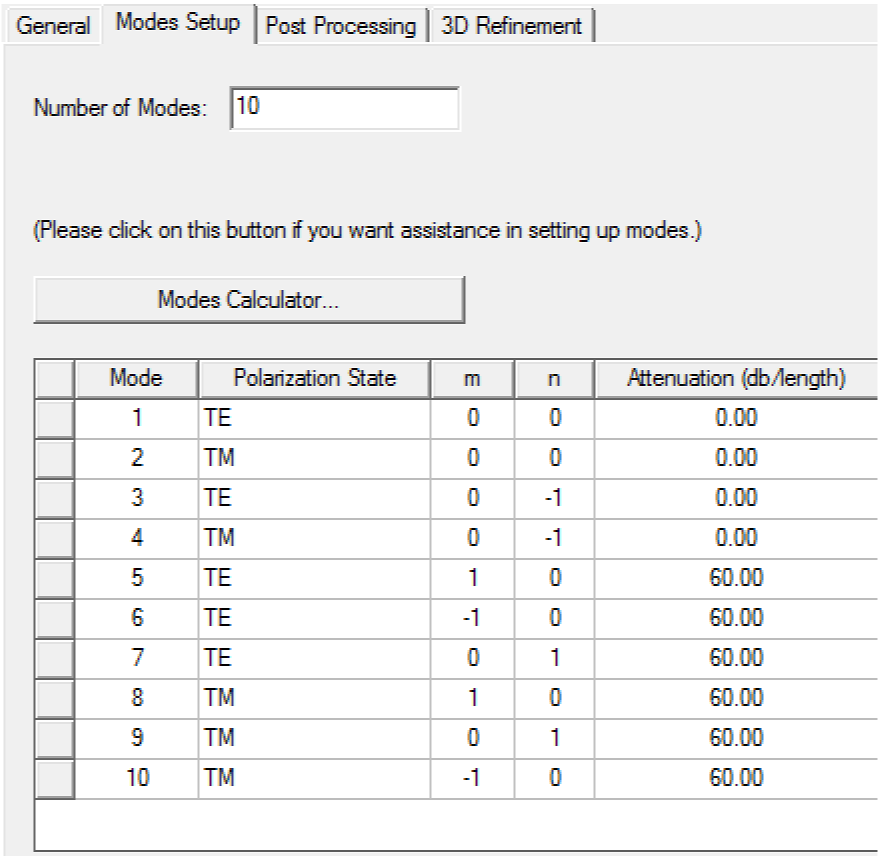
   The table gets trimmed to only 4 modes. The reasoning for trimming the modes is given in the next section.

10. Click **OK**.

11. Right-click **Parametric-Setup1** and select **Analyze**.

    HFSS runs the Parametric Sweep.

    Note This will take some time.



Mode	Polarization State	m	n	Attenuation (db/length)
1	TE	0	0	0.00
2	TM	0	0	0.00
3	TE	0	-1	0.00
4	TM	0	-1	0.00
5	TE	1	0	60.00
6	TE	-1	0	60.00
7	TE	0	1	60.00
8	TM	1	0	60.00
9	TM	0	1	60.00
10	TM	-1	0	60.00

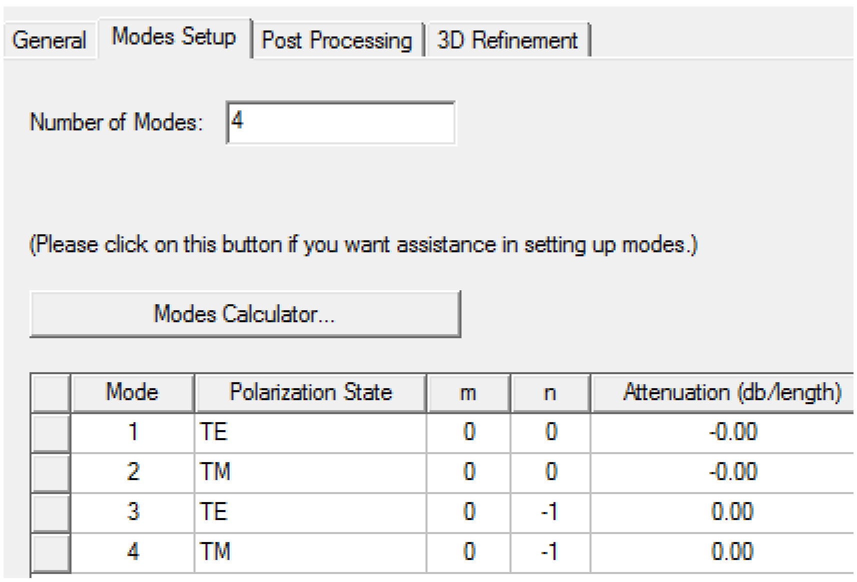
Figure 2-35 Floquet Port dialog open on Modes Setup

Mode Selection

In the previous section, you set up the modes using the **Modes Calculator** which resulted in the table in Figure 35. This section describes the table and the criteria for mode selection. The attenuation (db/length) depends upon the scan angle you set in the **Mode Table Calculator**. Refer to Figure 35 and note the following details about the results.

- The original pair of specular modes, TE_{00} and TM_{00} remain at the top of the list.
- The attenuation for each of TE_{00} and TM_{00} is 0 dB meaning that these modes are propagating unattenuated in at least one scan direction among those selected in "Mode Table Calculator" on the previous page. .
- A second pair of modes, the TE_{0-1} and TM_{0-1} are also propagating unattenuated in at least one scan direction.
- The six remaining modes experience attenuation of 60 dB per meter over the set of specified scan directions.

- Any Floquet modes generated within the 3D portion of the model and propagating toward the Floquet port must be accounted for in the modes table. Since the first four listed modes reach the Floquet port unattenuated, we will retain them.
- The remaining modes, with large non-zero attenuation can be excluded from the table. Eliminating unnecessary modes increases simulation efficiency and eases interpretation of results. The next point provides a quantitative justification for the omission of the modes with non-zero attenuation.
- Because the length of the unit cell is 1.25 meters, any of the last six modes that are generated at the rectangular aperture experience a minimum attenuation of **1.25 * 60 = 75 dB** by the time they reach the Floquet port plane. In most simulations this is small enough so that you can neglect such modes.



Mode	Polarization State	m	n	Attenuation (db/length)
1	TE	0	0	-0.00
2	TM	0	0	-0.00
3	TE	0	-1	0.00
4	TM	0	-1	0.00

Figure 2-36 Selected Modes

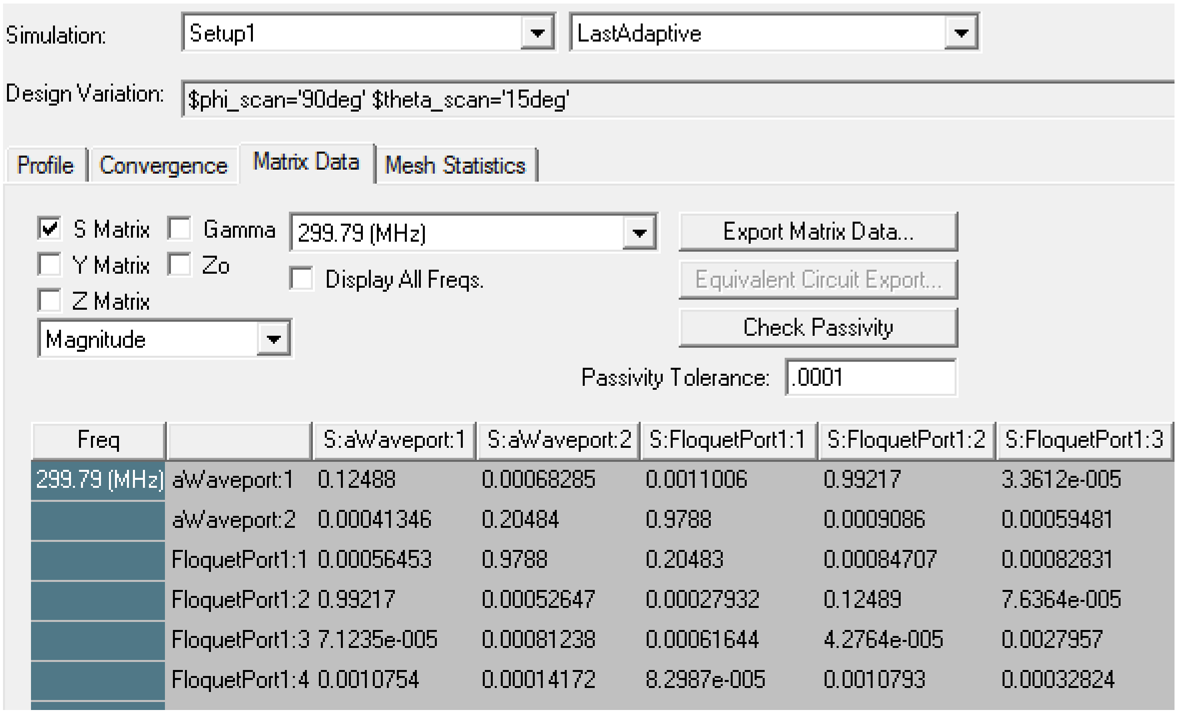
If you compare the last two figures, the list is trimmed from the bottom up and now only the TE_{00}, TM_{00}, TE_{0-1}, and TM_{0-1} remain.

Note You can increase the modes by entering a number more than 4 for the exact number to reappear in the list. Remember to always use the minimum number of modes to ensure simulation efficiency.

Viewing the Results of Parametric Sweep

Once the simulation is over, the S-matrix elements as a function of scan angle can be examined in the **Matrix Data** panel or the Reporter. In Figure 37, the design variation is set for **$phi_scan**=*90 deg* and **$theta_scan**=*15 deg*. You can click the ellipsis to the right of the **Design Variation** field and select *15 deg* for **$theta_scan** from the **Set Design Variation** dialog box.

The first column of the S-matrix consists of the array element active reflection coefficient and the coupling of the Wave port to the various Floquet modes. By viewing the matrix elements for a sampling of scan angles, it will be immediately noted that the couplings to the TE_{00} and TE_{0-1} modes are very small.



Freq		S:aWaveport:1	S:aWaveport:2	S:FloquetPort1:1	S:FloquetPort1:2	S:FloquetPort1:3
299.79 (MHz)	aWaveport:1	0.12488	0.00068285	0.0011006	0.99217	3.3612e-005
	aWaveport:2	0.00041346	0.20484	0.9788	0.0009086	0.00059481
	FloquetPort1:1	0.00056453	0.9788	0.20483	0.00084707	0.00082831
	FloquetPort1:2	0.99217	0.00052647	0.00027932	0.12489	7.6364e-005
	FloquetPort1:3	7.1235e-005	0.00081238	0.00061644	4.2764e-005	0.0027957
	FloquetPort1:4	0.0010754	0.00014172	8.2987e-005	0.0010793	0.00032824

Figure 2-37 Solutions: Matrix Data

Generate Plots

This section describes how to generate reflection and transmission plots. To create a report showing the (active) reflection and the TM transmission magnitudes plotted as a function of scan angle:

1. Right click **Results** in the Project tree, and from the short cut menu select **Create Modal Solution Data Report> Rectangular Plot**.

   The **Report** dialog box appears.
2. Set the fields as shown in following figure.

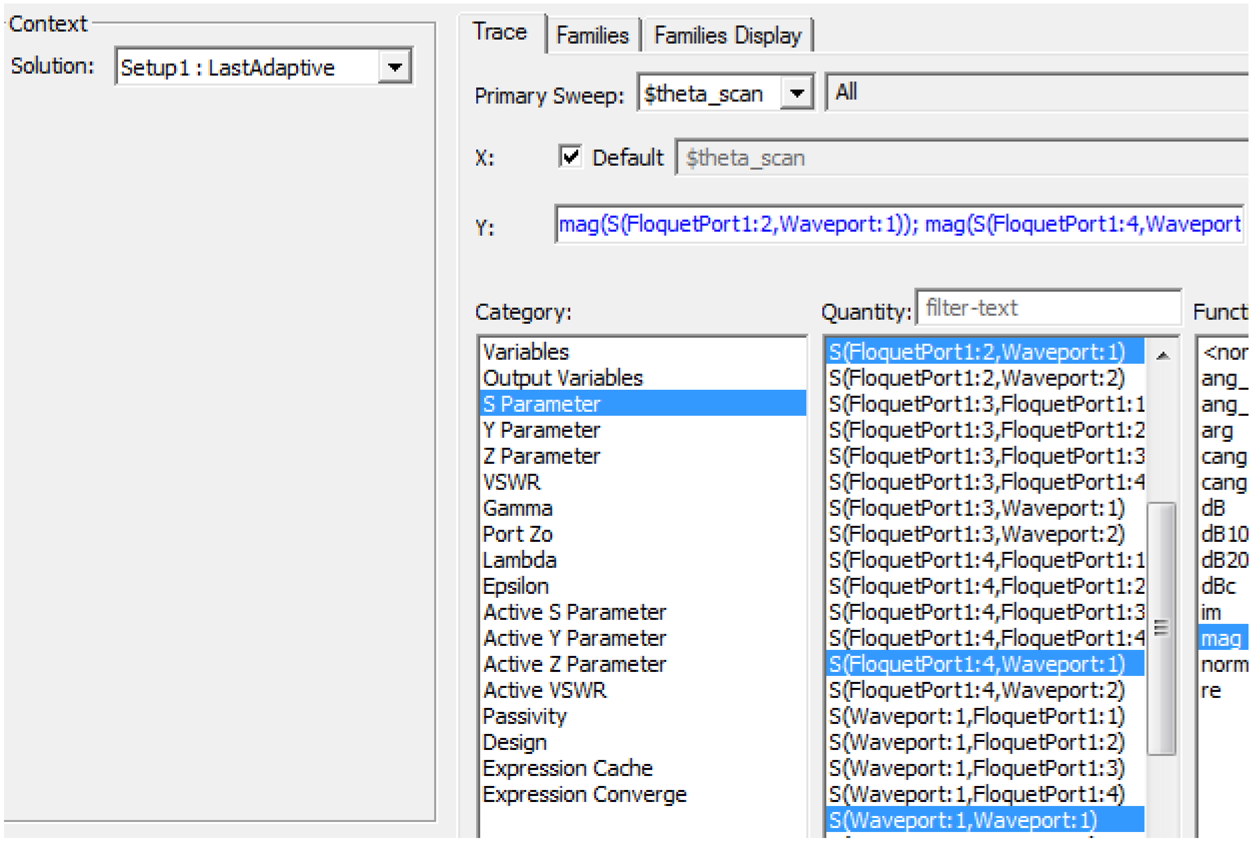


Figure 2-38 Report dialog box

3. Click **New Report**.

   The new report is displayed.

   Note The plot should resemble the one in following figure.

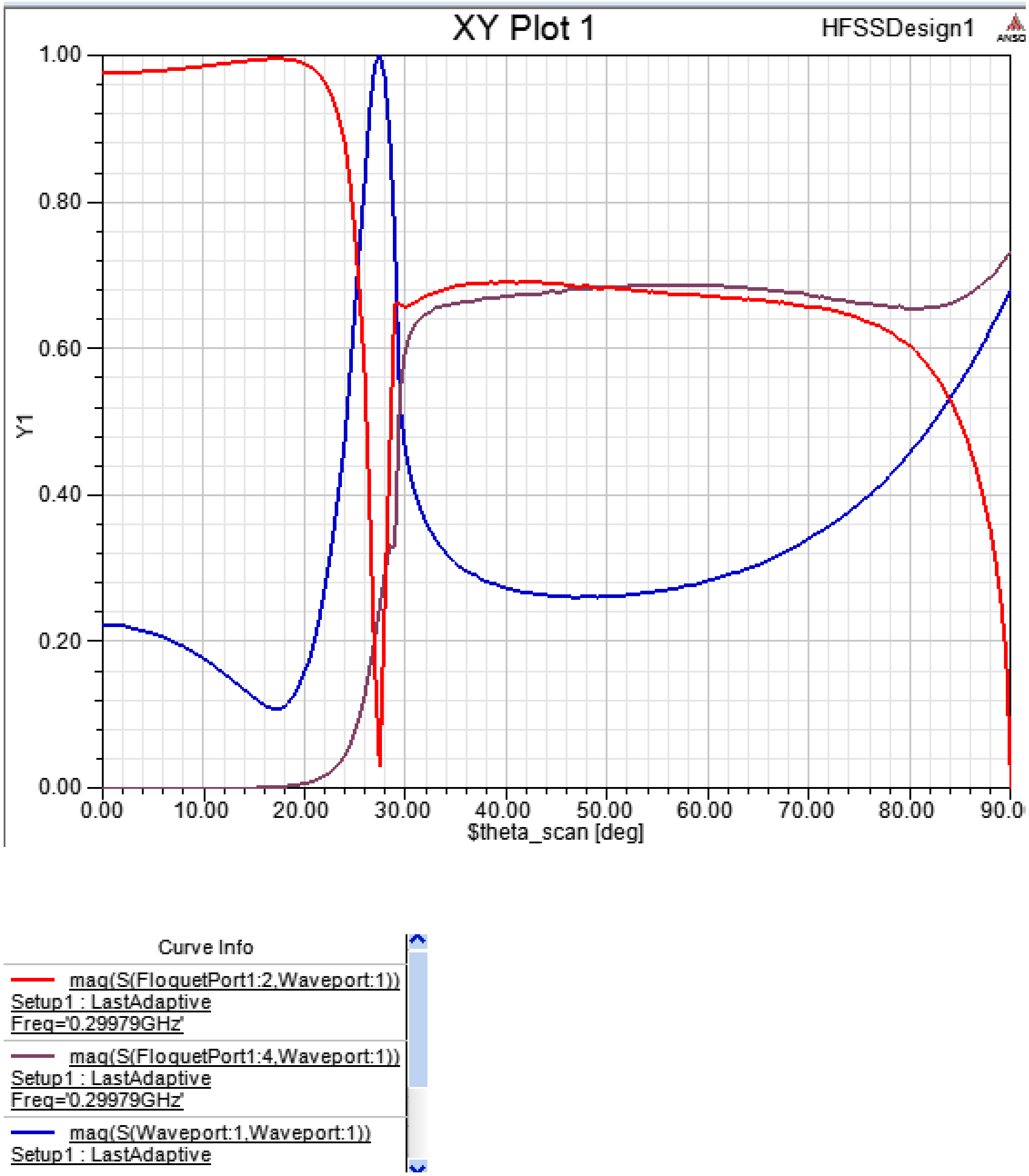


Figure 2-39 the plot and the legend

Note Observe that the active reflection coefficient shows the scan blindness condition at 27.5 degrees. Also note that the TM_{0-1} mode becomes propagating at about 30 degrees and represents the onset of a grating lobe. Beyond this point, power is about equally split between the TM_{00} and TM_{0-1} modes until the endfire condition is approached.

3 - Frequency Selective Surface Model

This chapter contains the following topics:

- Sample Project - FSS
- Set Unit and Solution Type
- Create the Unit Cell Model
- Create the Rhombic Sheet
- Transform 2D Sheet into 3D Cell Object
- Position the Rhombus Appropriately
- Create the Aperture
- Move the Aperture to the Rhomboid
- Assign Master and Slave Boundaries
- Assign the Perfect E-Boundary
- Assign the Floquet Ports
- Setup the Analysis
- Add Frequency Sweep
- Run the Simulation
- Generate the Reports
- Phase Transmission Vs Frequency
- S-Parameters Vs Frequency
- Port Field Display for Modes

Sample Project - FSS

You can construct unit cells for frequency selective surface (FSS) simulations using linked boundaries and two Floquet ports, with one port above the plane of the structure and the other port under it. The applied excitations are the Floquet modes themselves, usually one or both specular modes. As a direct result of the field solution, the reflection and transmission properties of the FSS are cast in terms of the computed S-matrix entries interrelating the Floquet modes.

This is somewhat in contrast to the simulation setup when PMLs or radiation boundaries are used to terminate the unit cell. In these cases, in addition to the boundary setup, one or more incident waves are defined separately as the excitation. The transmission and reflection properties of the FSS are then automatically computed for the user as a post-processing operation on the field solution.

We will consider a conducting screen containing a rhombic lattice of circular apertures. The lattice geometry is shown in "The Lattice Geometry" on the next page. in which, a pair of lattice vectors are drawn in blue. The angle between the lattice vectors is 60 degrees.

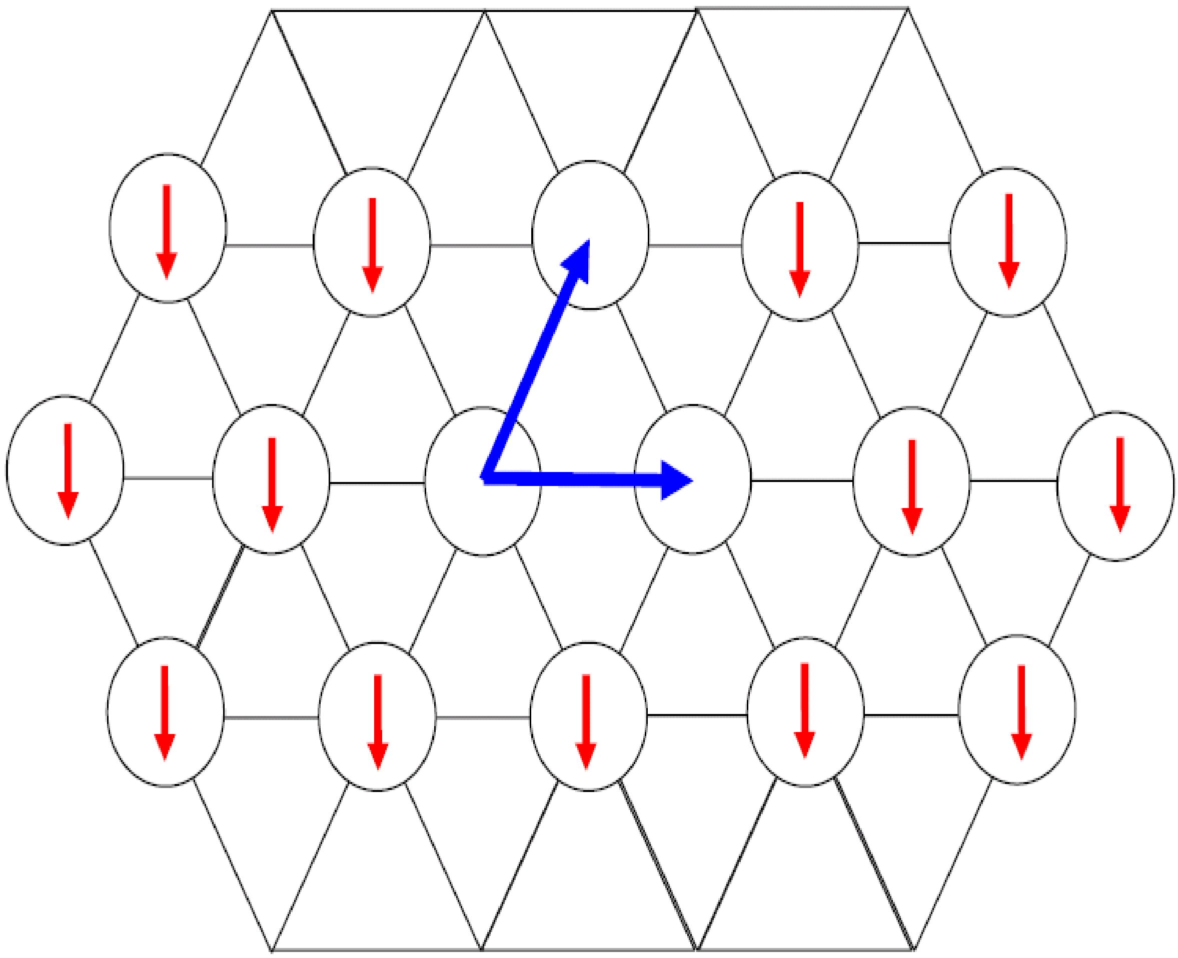


Figure 3-1 The Lattice Geometry

Consider a plane wave incident normally on the screen with polarization aligned as indicated by the red arrows in the figure. The transmission loss magnitude and phase as a function of frequency are the quantities of interest. The frequency band is 8 to 20 GHz.

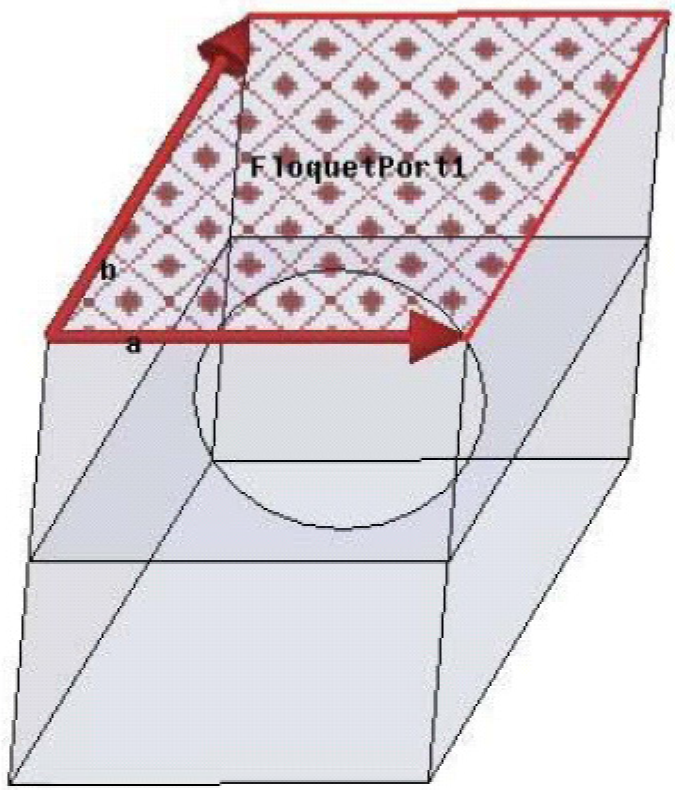


Figure 3-2 Unit cell for rhombic lattice

Set Units and Solution Type

Set the units and solution type for this project before you design the model.

1. Open a new Project and name it *rhombicArray*.
2. Click **Modeler**>**Units** and set the units to *cm*and click **OK.**
3. Insert an HFSS design type.
4. Click **HFSS**>**Solution Type**.
5. Select **Modal**.
6. Click **OK**.

Create the Unit Cell Model

A simple unit cell model has been created and is shown in "Unit cell for rhombic lattice" above The lengths of the side walls are 1.73 cm, the circular aperture diameter is 1.2 cm, and the cell is 4 cm high. The unit cell clearly is a rhombic object. The section below describes exactly how to draw this rhombic object.

Create the Rhombic Sheet

1. Click **Draw>Line**, click an arbitrary point to set the starting point, and then, click three more arbitrary points.
2. Return to close the line by clicking again on the starting point.
3. Right click and select **Done** from the short-cut menu.

   The **Properties** dialog box appears.
4. Click **OK**.

   This creates an initial Polyline object.
5. Double-click the first **CreateLine** on the history tree.

First CreateLine Properties dialog box

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	0 ,0 ,0	cm	0cm , 0cm , 0cm
Point2	0.865 ,1.4982 ,0	cm	0.865cm , 1.4982cm , 0cm

Figure 3-3 Segment tab (1)

6. Enter Point1 and Point2 co-ordinates as in Figure 3.

   This moves the first Polyline segment, and gives it the dimensions required.
7. Repeat this process with the next three CreateLines in the history tree, and edit their Point1 and Point2 co-ordinates as in figures 4 through 6.

   This creates the rhombic sheet for the rhombic cell. You may want to make a copy of the object as a first step in creating the aperture object.

Second CreateLine Properties dialog box

Name	Value	Unit	Evaluated Value
Segment Type	Line		
Point1	0.865 ,1.4982 ,0	cm	0.865cm , 1.4982cm , 0cm
Point2	2.595 ,1.4982 ,0	cm	2.595cm , 1.4982cm , 0cm

Figure 3-4 Segment tab (2)

Third CreateLine Properties dialog box

	Name	Value	Unit	Evaluated Value
	Segment Type	Line		
	Point1	2.595 ,1.4982 ,0	cm	2.595cm , 1.4982cm , 0cm
	Point2	1.73 ,0 ,0	cm	1.73cm , 0cm , 0cm

Figure 3-5 Segment tab (3)

Fourth CreateLine Properties dialog box

	Name	Value	Unit	Evaluated Value
	Segment Type	Line		
	Point1	1.73 ,0 ,0	cm	1.73cm , 0cm , 0cm
	Point2	0 ,0 ,0	cm	0cm , 0cm , 0cm

Figure 3-6 Segment tab (4)

Transform 2-D Sheet into a 3D Cell Object

We recommend, you make a copy of the 2-D sheet because you will need it when you create an aperture later.

1. Click **Polyline1** in the tree to highlight the 2-D sheet.
2. Do **Ctrl+C**.
3. Click **Polyline** to highlight it again.
4. Click **Draw**>**Sweep**>**Along Vector**.
5. Input the first point of the sweep vector in the status bar as follows: **X** = *0*, **Y** = *0*, **Z** = *0.*
6. Input the second point of the sweep vector as follows:
   dx = *0*, **dy** = *0*, **dz** = *4*

   The **Sweep Along Vector** dialog box appears.

Figure 3-7 Sweep Along Vector

7. Verify that the **Draft angle** = *0* and **Draft type** = *Round* and click **OK**.

   The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	SweepAlongVector		
	Coordinate Sys...	Global		
	Vector	0 ,0 ,4	cm	0cm , 0cm , 4cm
	Draft angle	0	deg	0deg
	Draft type	Round		
	Suppress Com...	☐		

Figure 3-8 Properties dialog box

8. Verify that your settings match the ones in "Properties dialog box" above. .
9. Click **OK**.

   This completes the sweep converting the 2 D sheet object into a 3-D solid.

Position the Rhombus Appropriately

1. Do **Ctrl+D** to fit the object into the 3-D Modeler window.
2. Select the object.
3. Click the **Move** icon on the toolbar.

   This lets you enter values in the status bar.
4. Input the reference point of the move vector:
   - X = 0, Y = 0, Z = 0.
5. Input the target point of the move vector:
   - dx = 0, dy = 0, dz = -2.0
6. Press **Enter**.

   The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	Move		
	Coordinate Sys...	Global		
	Move Vector	0 ,0 ,-2	cm	0cm , 0cm , -2cm
	Suppress Com...	☐		

Figure 3-9 Move Vector properties

7. Click **OK**.

   This moves the rhomboid to the proper position.

Create the Aperture Object

If the copy of the Polyline that you created before transforming the sheet to a solid, still exists in the computer memory, then if you go to **Edit** and click **Paste**, the sheet will appear in the center of the 3-D object. If it does not exist, then, repeat Steps 1 through 7 from the section “Create Rhombic Sheet.” Now perform the following steps to create the aperture.

1. Click **Draw>Circle**.
2. Click the center of the rectangle to produce a round dot under the cursor.
3. Drag the cursor towards the edge of the rectangle and click to draw the circle.

   The Properties window opens.
4. Click **OK**.
5. Double-click **CreateCircle** on the project tree to open the **Properties** dialog box and set the radius as 0.6 cm.

Name	Value	Unit	Evaluated Value
Command	CreateCircle		
Coordinate Sys...	Global		
Center Position	1.2975 ,0.7491 ,0	cm	1.2975cm , 0.74...
Axis	Z		
Radius	0.6	cm	0.6cm
Number of Seg...	0		0

Figure 3-10 Circle properties

6. Verify that the circle settings are the same as in "Circle properties" above. .
7. Select both the circle and the rectangle (Polyline2), and click **Modeler>Boolean>Subtract**.

   The **Subtract** dialog box appears.

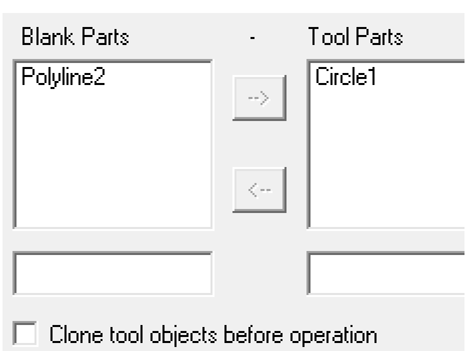


Figure 3-11 Subtract dialog box

8. Verify that Polyline2 is in the **BlankParts** and Circle1 in the **ToolParts**.
9. Click **OK** to close the dialog box.

   The circle is subtracted from the rectangle to create the aperture.

Move the Aperture to the Rhomboid

To be sure that the placement of the aperture inside the rhomboid object is accurate, you can move the aperture and enter the appropriate values in the **Properties** dialog box for the move vector.

1. Select the aperture and click **Move** from the toolbar.
2. Drag the aperture object inside the rhomboid.
3. Double-click **Move** on the project tree.

   The **Properties** dialog box for the move vector opens.
4. Enter *0*, *0*, *0* in the **Move Vector** field.

Name	Value	Unit	Evaluated Value
Command	Move		
Coordinate Sys...	Global		
Move Vector	0 ,0 ,0	cm	0cm , 0cm , 0cm
Suppress Com...	☐		

Figure 3-12 Move Vector properties

Assign Master and Slave Boundaries

You have drawn the geometry for the FSS and now you can assign the Master and Slave boundaries to the rhomboid object as follows. Refer to "The Master and Slave boundaries applied" on the facing page. to get a sense of the faces where the U and V vectors are drawn.

1. Hit **F** to enter **FaceSelection** mode.
2. Select the face shown in following figure, and click **HFSS>Boundaries>Assign>Master.**

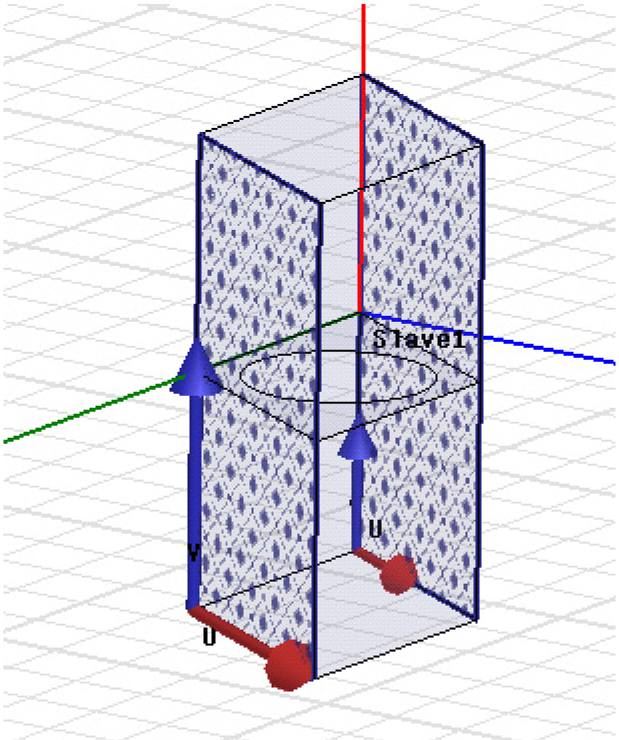


Figure 3-13 The Master and Slave boundaries applied

The **Master Boundary** dialog box appears.

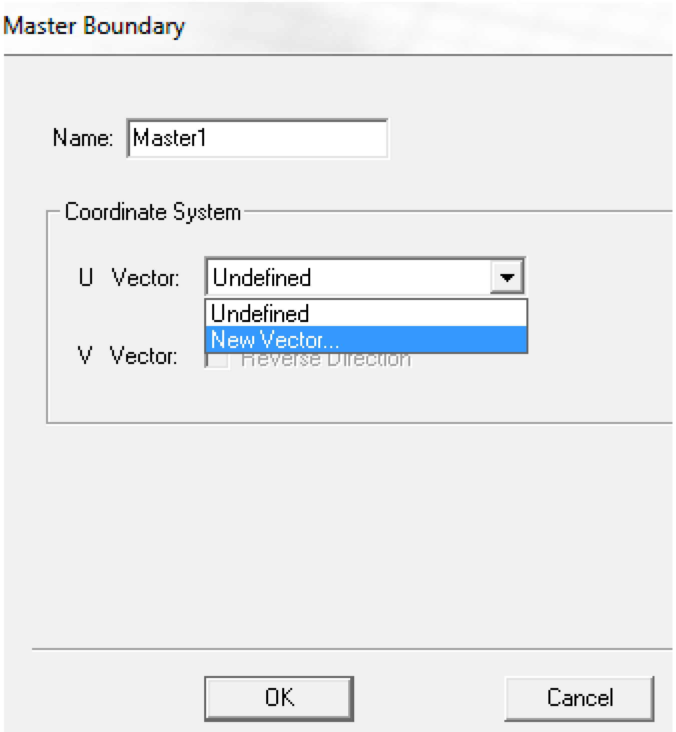


Figure 3-14 Master Boundary dialog box

3. Leave **Name** as *Master1*.
4. Select **New Vector** from the drop-down menu for **U**.

   The **Measure** dialog box and a the **Create Line** prompt appear.
5. Click the lower left corner as the starting point on the selected face, and drag the cursor to the right corner and click.

   U-vector is drawn, the **V-vector** points away from the box, and the **Master Boundary** dialog box appears again.
6. Check **Reverse direction** for the **V-vector**, click **OK** to close the dialog box.
7. Select the opposite face.
8. Click **HFSS>Boundaries>Assign Slave.**

   The **Slave** dialog box appears.

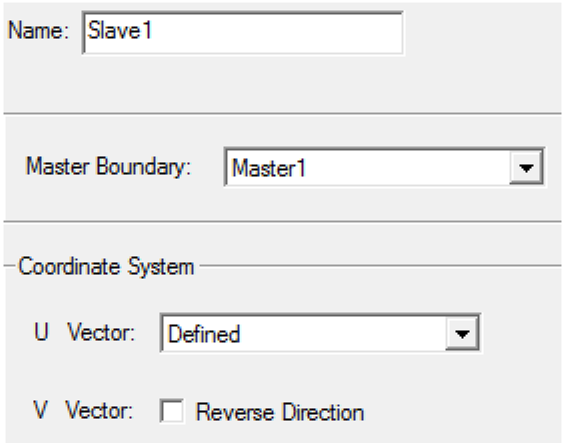


Figure 3-15 Slave dialog box after Slave is applied

9. Select *Master1* as the **Master Boundary** and *New Vector* for **U Vector**.
10. Draw the U Vector as shown.

    Note The U-Vector of the Slave1 will be along the same direction as the U of Master1 (from the bottom right-most corner of the face to the leftmost corner.)

11. Leave the scan angles at 0 each for Phi and Theta and click **Finish**.

    The Master1 and Slave1 boundaries are created.

12. Repeat the procedure for the Master2 and Slave 2 boundaries, as shown.

    Note Make sure you reverse the direction of the V vector as needed. The tail of the U-vector of Master2 coincides with the head of the U of Master1. Remember that the U-vector of Slave2 must be along the same direction as that of Master2.

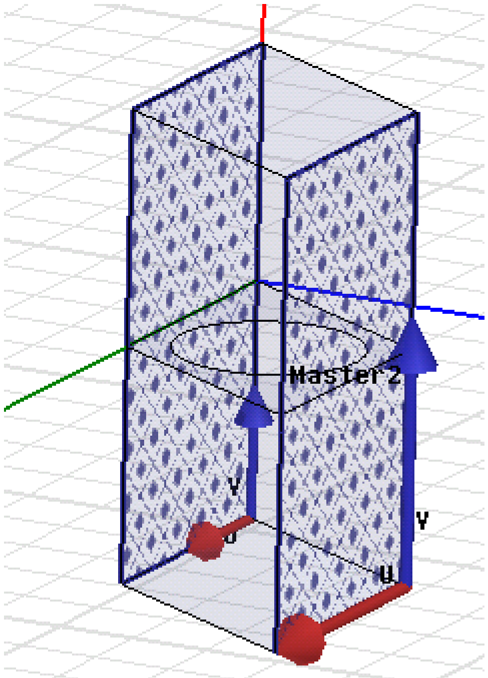


Figure 3-16 Master2 and Slave2

Assign the Perfect E Boundary

You will now assign a Perfect E boundary to the aperture.

1. Hit **O** to enter Object Selection mode.
2. Select the aperture.
3. Select **HFSS>Boundaries>Assign>Perfect E**.
4. Leave the **Name** as *PerfE1*, **Infinite Ground Plane** deselected and click **OK** to close the dialog box.

   The PerfE1 boundary appears on the aperture as shown in following figure.

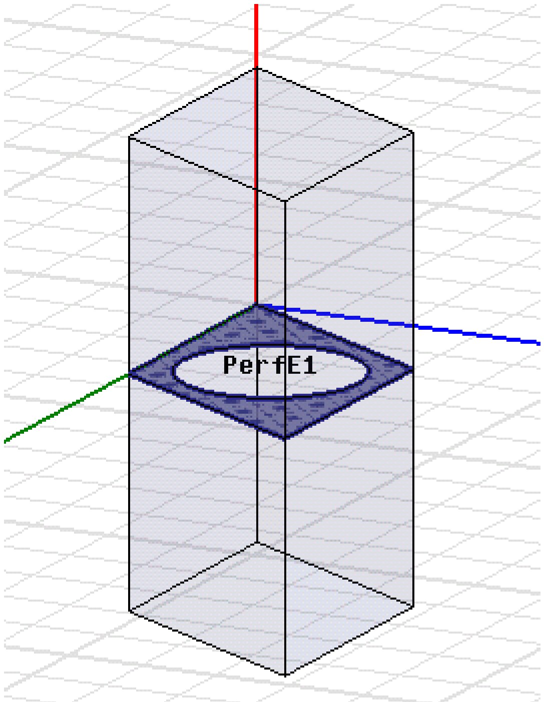


Figure 3-17 PerfE1 applied

Assign the Floquet Ports

You are all set to assign the Floquet ports to the top and bottom faces of the model.

1. Select the top face of the model, and click **HFSS>Excitations>Assign>Floquet Port**

   The Floquet Port wizard appears.

2. For the Lattice Coordinate System, from the drop down menu for **A direction**, select *New Vector*.

   The Measure Data dialog appears, and the Create Line dialog tells you to draw the **A** lattice vector on the plane of the face.

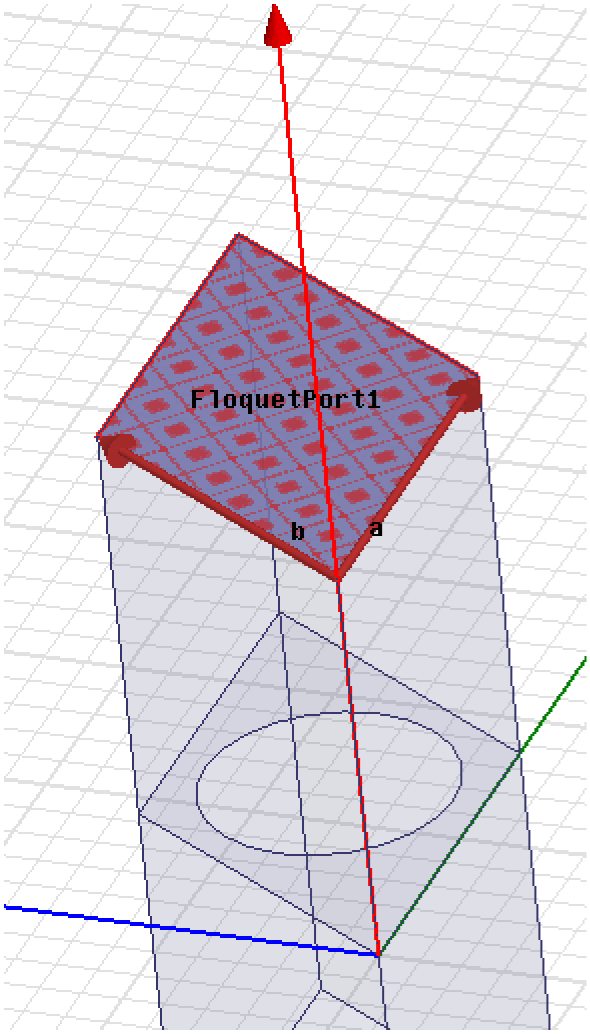


Figure 3-18 Floquet Port applied

3. Draw the “**a”** vector parallel to the X axis and then, click.

   Note Click the corner along the Z-axis first and then, draw the **“a**” vector.

   The Floquet Port dialog box appears.

   Note When you make the second click to complete the vector, the **Measure Data** and **Create line** dialogs disappear, and the **Floquet Port** wizard reappears, showing that the a vector is Defined.

4. Select *New Vector* from the **B Direction** drop-down menu and draw the **“b”** vector as shown.

   Note The “**b**” vector should be perpendicular to “**a**” and parallel to **Y** axis.

   The “b” vector is drawn and the Modes Setup tab opens.

5. Click **Modes Calculator** for assistance in setting up the modes.

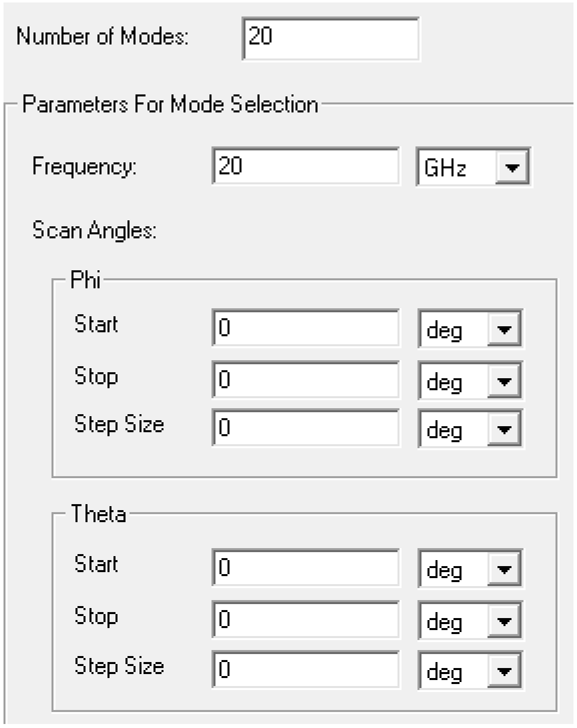


Figure 3-19 Mode Table Calculator

The Modes calculator appears.

6. Enter the settings as shown in "Mode Table Calculator" above. .
7. Click **OK** to close the Calculator.

This fills out the table with calculated values.

Mode	Polarization State	m	n	Attenuation (db/length)
1	TE	0	0	0.00
2	TM	0	0	0.00
3	TE	-1	0	1.15
4	TE	1	0	1.15
5	TE	-1	-1	1.15
6	TE	1	1	1.15
7	TM	1	1	1.15
8	TM	1	0	1.15
9	TM	-1	-1	1.15
10	TM	-1	0	1.15
11	TE	0	-1	1.16
12	TE	0	1	1.16
13	TM	0	-1	1.16
14	TM	0	1	1.16
15	TE	2	1	51.53
16	TE	-2	-1	51.53
17	TM	2	1	51.53
18	TM	-2	-1	51.53
19	TE	1	-1	51.53
20	TE	-1	1	51.53

Figure 3-20 Calculated Attenuation values

8. Re-enter *14* in the **Number of Modes** field.

   Note The reasoning for trimming the modes is given in the next section.

9. Click **Next**.

   The **Post Processing** dialog box appears.

10. Check the box for **Deembed**, and specify the **Distance** as *2.0* cm.

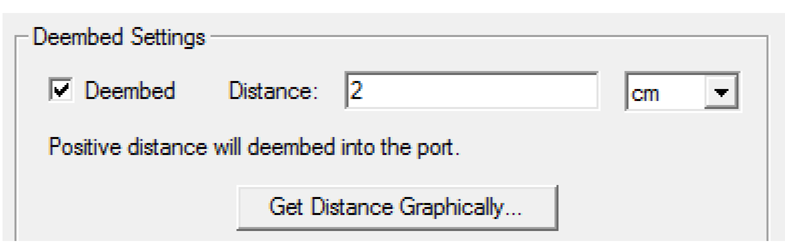


Figure 3-21 Post Processing

11. Click **Next** for the **3D Refinement** page.
12. For Mode 1, click the **Affects Refinement** check box.
13. Click **Finish**.

    The first Floquet port appears under the Excitations icon in the Project tree.
14. Select the bottom face of the model, and repeat the process to setup the second Floquet port.

    Note Verify that the information from the first Floquet port is copied over.

Omission of Modes with Large Attenuation

In the previous section, we trimmed the table (in Figure 20) from 20 to 14. Since the attenuation for more than 14 modes is significantly high, we eliminated those modes. For Deembed distance = 2cm, the total attenuation from the aperture to the port = (51.53db/cm)*(2cm) = 103.06db. This is very large and so we neglect modes through 15 to 20.

Setup the Analysis

1. Right click **Analysis** in the project tree, and select **Add Solution Setup**.

   This opens the **Solution Setup** dialog.

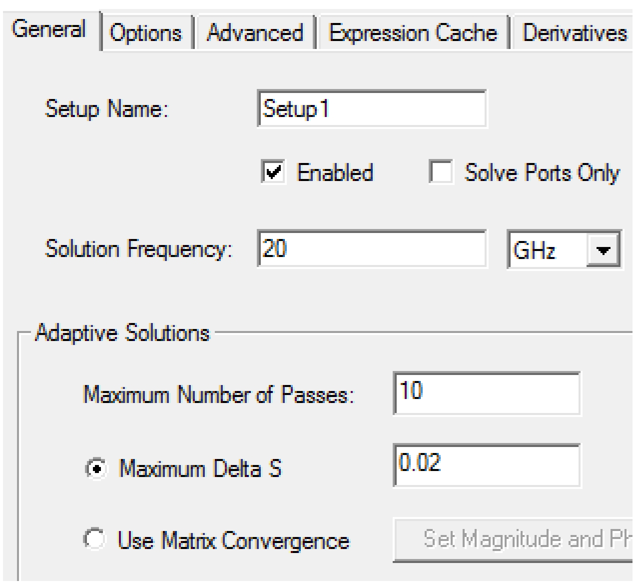


Figure 3-22 Solution Setup

2. Enter the settings as shown in "Solution Setup" above. .
3. Click **Options** and enter the setting as shown in following figure.

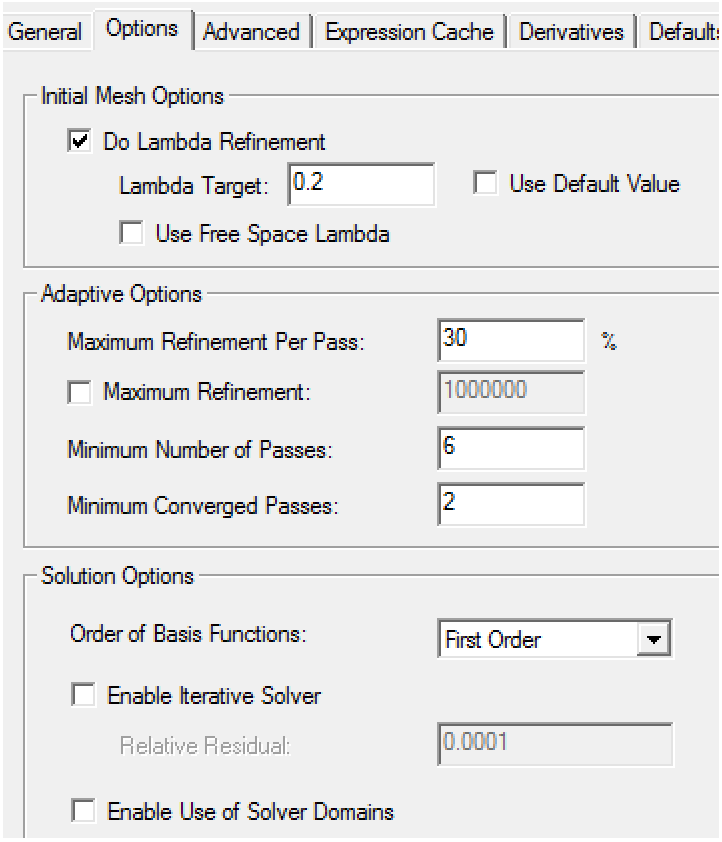


Figure 3-23 Options tab

4. Click **OK**.

Add Frequency Sweep

1. Right-click **Setup1** in the Project tree, and select **Add Frequency Sweep**.
   The **Edit Frequency Sweep** dialog box appears.

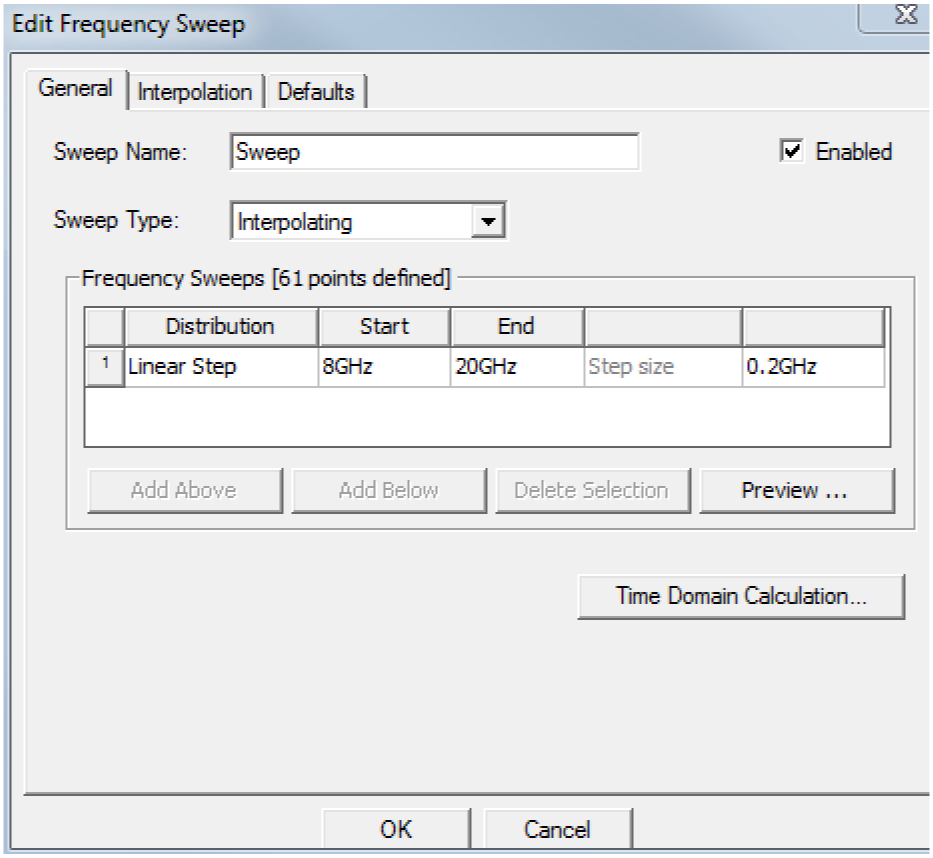


Figure 3-24 Edit Frequency Sweep dialog box

2. Enter the field settings as shown in "Edit Frequency Sweep dialog box" above. .
3. Click **Interpolation** and set fields as in following figure.

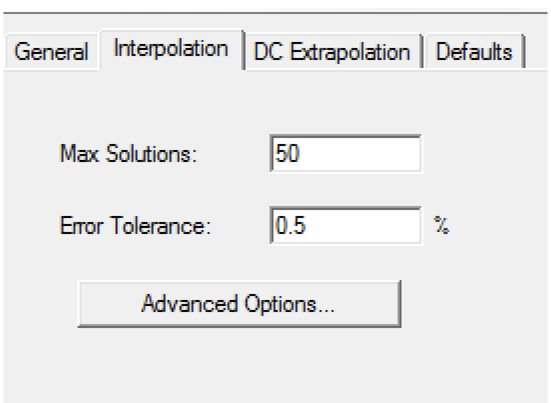


Figure 3-25 Interpolation tab

4. Click **Advanced Options**.

   Interpolating Sweep Advanced Options dialog appears.

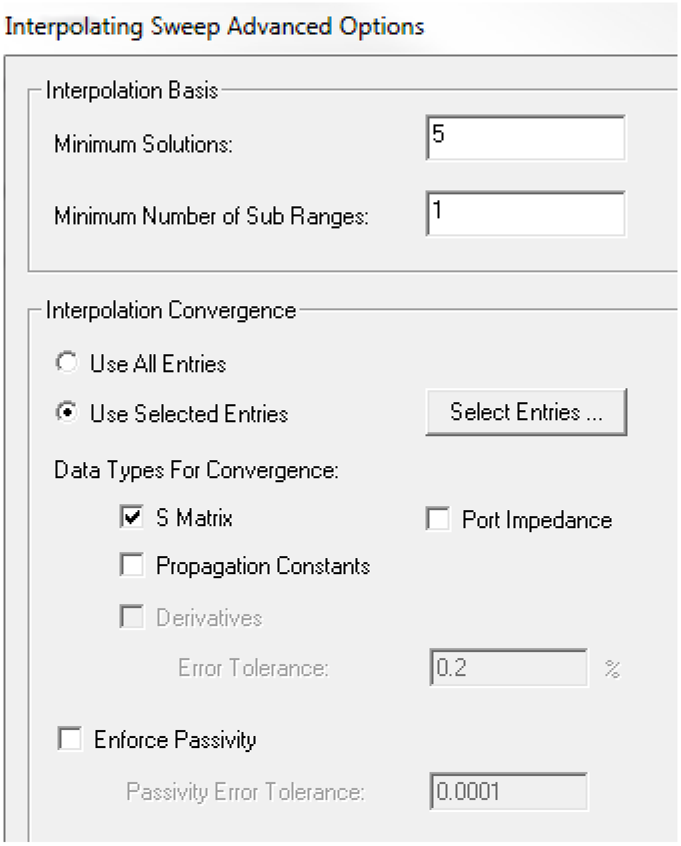


Figure 3-26 Interpolating Sweep Advanced Options

5. Enter the fields as shown in "Interpolating Sweep Advanced Options" above. .
6. Click **Select Entries**.

   The **Interpolation Basis Convergence** dialog box opens.
7. Select *All* for fields **Entry** and **Mode Selections**.
8. Scroll down to select the row **FloquetPort2:1**, and set the value in the **FloquetPort1:1** column on that row to *ON*.

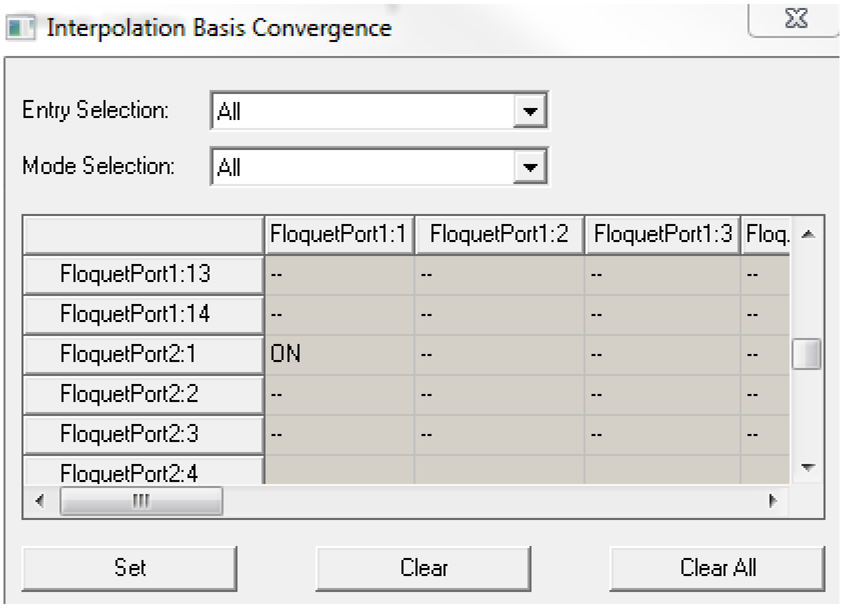


Figure 3-27 Interpolation Basis Convergence

9. Click **OK** to close the **Interpolation Basis Convergence** dialog box, **OK** to close the **Interpolating Sweep Advanced Options** dialog box, and **OK** to close the **Edit Sweep** dialog box.
10. Click **HFSS**>**Validation Check**.

    The validation check should yield correct results.

    Note For most HFSS projects, do not be alarmed if you see any warnings in the Message Manager window. Some of these warnings are informational messages only. They may not always require you to perform any action to deal with the messages. However, for these projects, the **Message Manager** must not display any error or warning messages; the **Validation Check** dialog box must show a tick near each of the listed options, in which case, you can analyze the design.
11. Right-click **Sweep** in the Project tree and click **Analyze** on the shortcut menu.

    The simulation starts and finishes after some time.

Generating Reports

Now that you solved your design, you can generate reports for further analysis of the transmission and reflection of the wave through HFSS.

Phase Transmission Vs Frequency

1. Right click **Results** and click **Create Modal Solution Data Report>Rectangular Plot** from the project tree.

   The **Report** dialog box appears.

2. The **Context** portion of the dialog box should resemble the entries in "Context part of the Report dialog box" below. .
3. Enter the fields for **Trace** as in "Trace part of the Report dialog box" below. so that the Y axis is populated with the expression for phase transmission.
4. Leave the **Default** value of **X** at *Freq*.

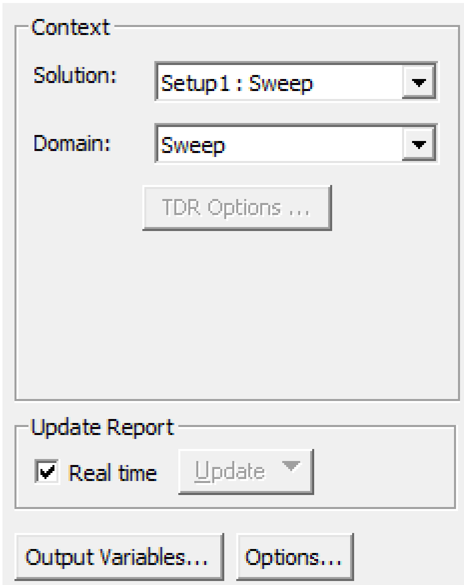


Figure 3-28 Context part of the Report dialog box

5. Click **New Report**.

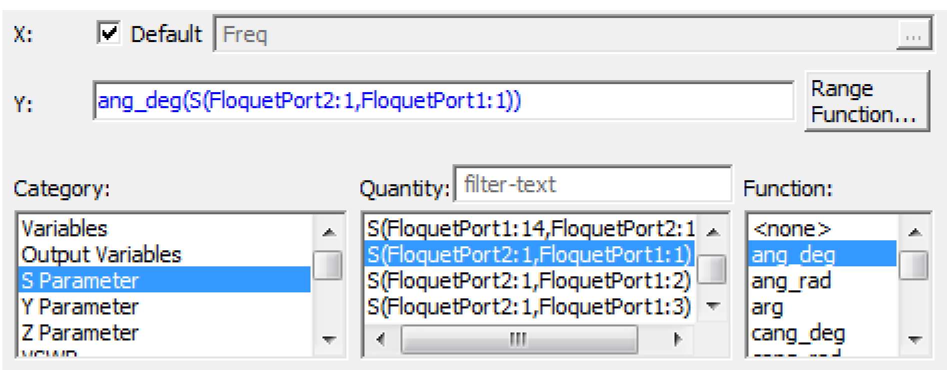


Figure 3-29 Trace part of the Report dialog box

HFSS generates the plot for Phase Transmission versus Frequency. See following figure. Clearly, from the plot we can infer that phase changes as the wave passes through the FSS.

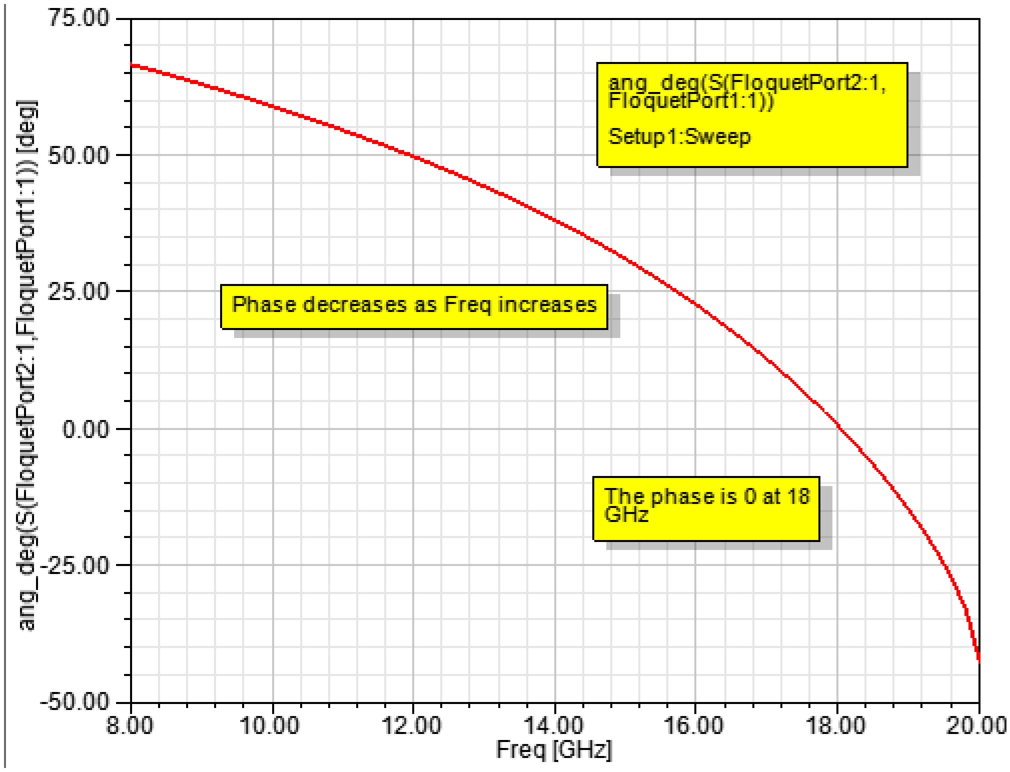


Figure 3-30 Phase Transmission Vs Frequency

S Parameters Vs Frequency

1. Right-click **Create Modal Solution Data Report>Rectangular Plot** from the project tree.
2. Select *S Parameter* from **Category**.
3. Set **Y**= *dB(S(FloquetPort1:1,FloquetPort1:1))* in the **Report** dialog box.
4. Set the **Function** as *dB*.
5. Click **New Report**.

   The transmission curve is generated.
6. Change **Y** to *dB(S(FloquetPort1:1,FloquetPort2:1))*

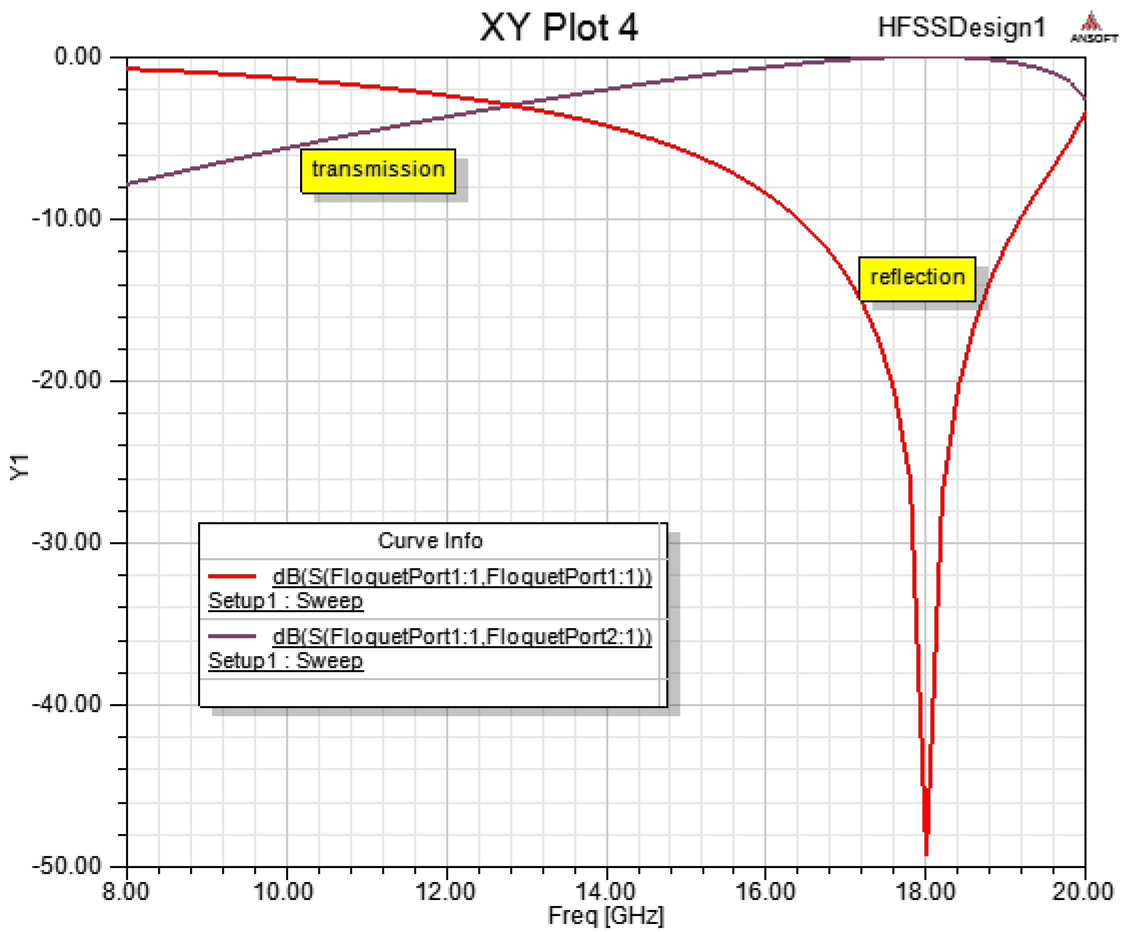


Figure 3-31 Transmission & Reflection Vs Frequency

7. Click **Add Trace**.

   The second curve is added to the existing trace. Note the strong transmission through the FSS at 18GHz.

Port Field Displays for Modes

If you double-click the Mode under Floquet Port1 or Floquet Port2 on the Project Tree you can view the field display for the corresponding mode selected.

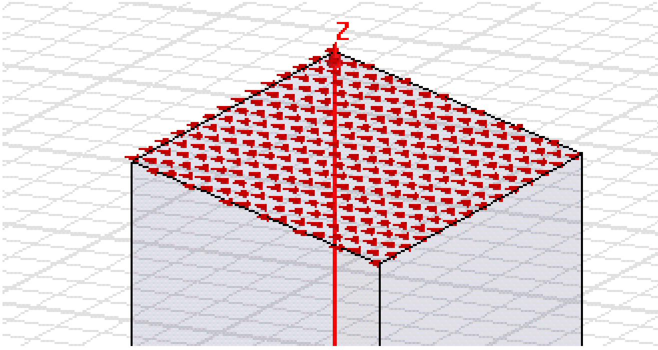


Figure 3-32 Mode 1 Port Display

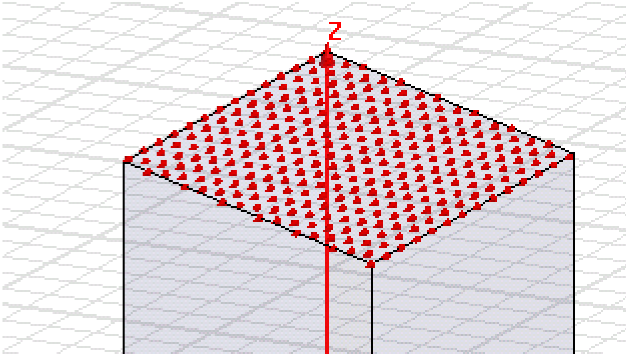


Figure 3-33 Mode 2 Port Field Display

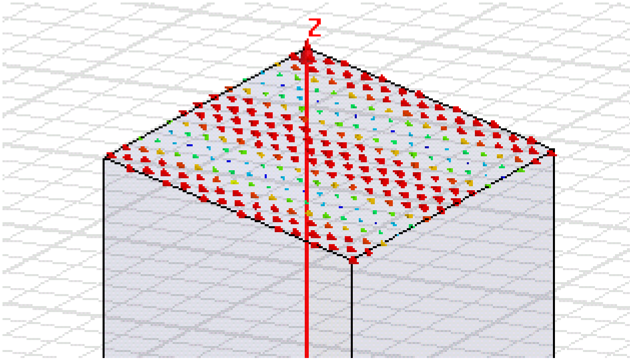


Figure 3-34 Mode 3 Port Field Display

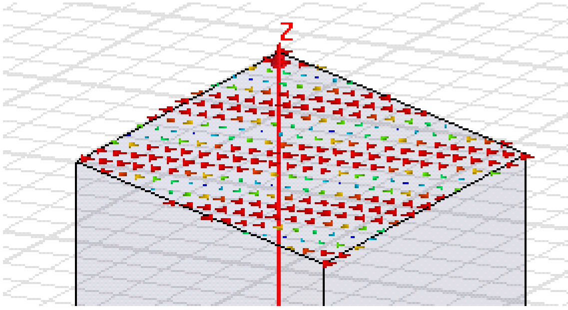


Figure 3-35 Mode 4 Port Field Display