



## Getting Started with HFSS: Floquet Ports



July 2007

The information contained in this document is subject to change without notice. Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2007 Ansoft Corporation. All rights reserved.

Ansoft Corporation  
225 West Station Square Drive  
Suite 200  
Pittsburgh, PA 15219  
USA  
**Phone:** 412-261-3200  
**Fax:** 412-471-9427

HFSS and Optimetrics are registered trademarks or trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.

New editions of this manual incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.


Edition	Date	Software Version
1	May 2007	11

## Conventions Used in this Guide

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

*Alternate methods or tips are listed in the left margin in blue italic text.*

- 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 to type the word **copy**, to type a space, and then to type **file1**.
  - 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**” means to type the word **copy**, to type a space, and then to type a file name.
- 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

### Ansoft Technical Support

To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, <http://www.ansoft.com>, click the **Contact** button, and then click **Support**. Phone numbers and e-mail addresses for the technical support staff are listed. You can also contact your Ansoft account manager in order to obtain this information.

All Ansoft software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

### Help Menu

To access online help from the HFSS menu bar, click **Help** and select from the menu:

- **Contents** - click here to open the contents of the online help.
- **Search** - click here to open the search function of the online help.
- **Index** - click here to open the index of the online help.

### Context-Sensitive Help

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

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

# Table of Contents

## 1. Introduction

Floquet Ports in HFSS .....	1-2
-----------------------------	-----

## 2. Antenna Array

Array antenna tutorial example .....	2-2
Create the Unit Cell Model .....	2-4
Assign the Master and Slave Boundaries .....	2-5
Assign the Wave Port .....	2-6
Assign the Floquet Port .....	2-8
Create the Project Variables .....	2-10
Setup the Analysis .....	2-11
Run Analysis and View Results .....	2-12
Parametric Sweep of Scan Angle .....	2-13
Prepare the Modes for the Parametric Sweep	2-14
Viewing the Results of the Parametric Sweep	2-17

## 3. Frequency Selective Surface Model

Frequency Selective Surface Example .....	3-2
Create the Unit Cell Model .....	3-4
Create the Rhombic Object .....	3-4
Create the Aperture Object .....	3-5
Move the Aperture to the Rhomboid .....	3-6

Assign the Master and Slave Boundaries . . . . .	3-7
Assign the Perfect E Boundary . . . . .	3-9
Assign the Floquet Ports . . . . .	3-9
Setup the Analysis . . . . .	3-11
Setup the Sweep . . . . .	3-12
Viewing the Results . . . . .	3-13

# 1

## Introduction

This *Getting Started* guide is written for HFSS users who are using the Floquet port feature in version 11 for the first time. This guide leads you step-by-step through creating, solving, and analyzing the results of models that use a Floquet port.

By following the steps in this guide, you will learn how to perform the following tasks in HFSS:

- ✓ Draw the geometric models.
- ✓ Modify the models' design parameters.
- ✓ Specify solution setting 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, at least one "open" boundary condition representing the boundary to infinite space is needed. Heretofore a PML or sometimes a radiation boundary has been used. The Floquet port is a new option.

The Floquet port is closely related to a Wave port in that a set of modes is used to represent the fields on the port boundary. The new modes are called Floquet modes. 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 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 version 11 release, 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

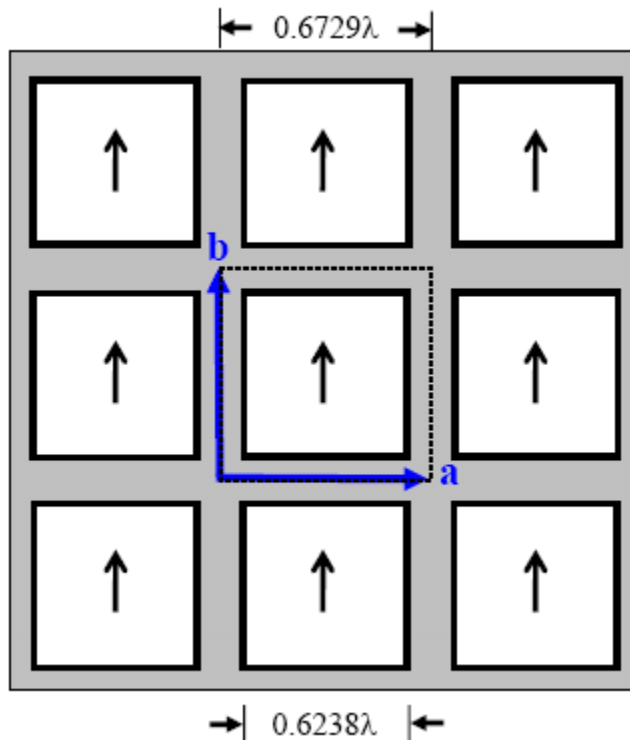
## Antenna Array

This chapter guides you through the creation and simulation of the antenna array model.

- ✓ Create the Unit Cell
- ✓ Assign the Boundaries
- ✓ Assign the Floquet Ports
- ✓ Setup the Analysis
- ✓ Run the Simulation
- ✓ Create the Reports

## Array antenna tutorial example

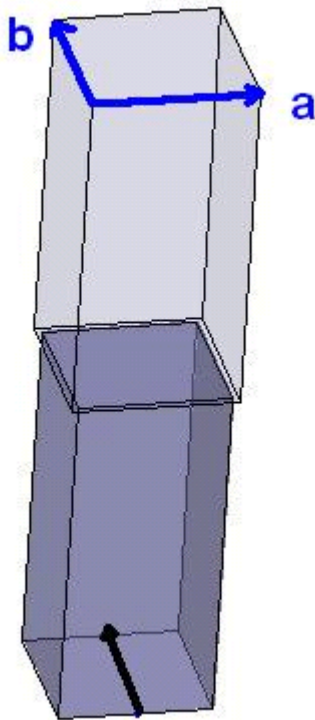
As a first illustration, consider the case of an infinite array of square aperture antennas in a conducting plane. The geometry of the array lattice is shown in Figure 1. The elements are fed from below the plane by square waveguide ports with dominant modal field aligned in the direction shown.



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  $a$  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 the figure using dotted lines.

Figure 2 depicts an HFSS model for the unit cell of the infinite array. The model consists of two boxes. The bottom 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.



**HFSS model of unit cell with lattice vectors**

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

## Create the Unit Cell Model

To create the unit cell model:

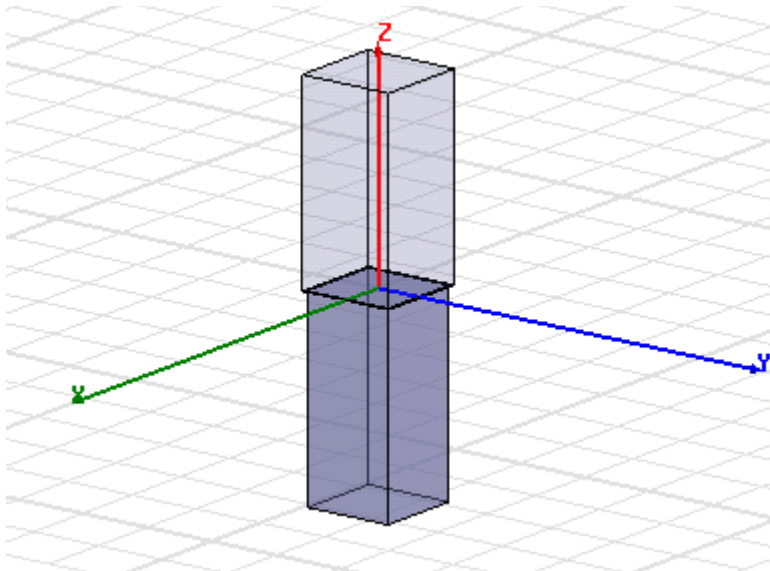
- 1** Open a Project and name it AGW.
- 2** Use **Draw>Box** to create an arbitrary box, and **Edit>Properties** as follows:

Position	-0.33645 , -0.33645 , 0 meter
X Size	0.6729 meter
Y Size	0.6729 meter
Z Size	1.4 meter

- 3** Select the newly created box, and edit the Transparency to 0.86

- 4** Use **Draw>Box** to create a second arbitrary box, and **Edit>Properties** as follows:

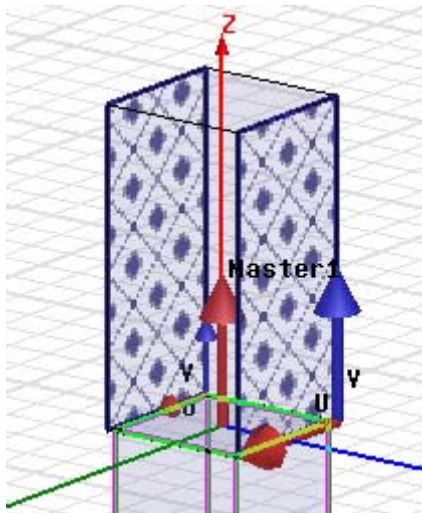
Position	-0.3119 , -0.3119 , 0 meter
X Size	0.6238 meter
Y Size	0.6238 meter
Z Size	-1.4 meter



## Assign the Master and Slave Boundaries

Assign the Master and Slave boundaries to the box object as follows.

- 1 Select the face of the upper box shown in the figure, and click **HFSS>Boundaries>Assign>Master...** or right click on Boundaries in the Project tree and select **Assign>Master...**



The Master Boundary dialog appears.

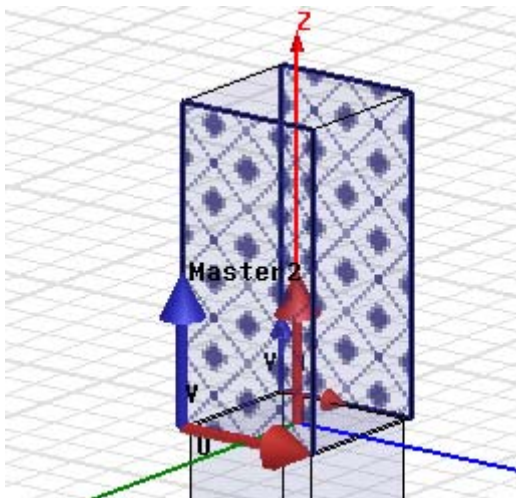
- 2 Accept the default name as Master1.
- 3 Click the drop down menu for the U vector, and click, New Vector.

The Measure dialog and a the Create Line prompt appears.

- 4 Draw the U vector of the coordinate system in the place shown on the face in the figure. Click on the lower right corner as the starting point, and drag the cursor to the left corner and click.
- 5 Click OK to close the dialog.
- 6 Select the opposite face, and **HFSS>Boundaries>Assign Slave...** or right click on Boundaries in the Project tree and select **Assign>Slave...**

The Slave dialog appears with the General Tab selected.

- 7** Select Master1 as the Master boundary.
- 8** Draw the U Vector as shown.
- 9** Check the Reverse direction dialog for the V vector.
- 10** Accept the other settings and OK to close the dialog.
- 11** Repeat the procedure for the Master2 and Slave 2 boundaries, as shown. The main difference is that for the Master2 boundary, you do check the Reverse direction dialog for the V vector, but you don't check it for the Slave2 boundary.

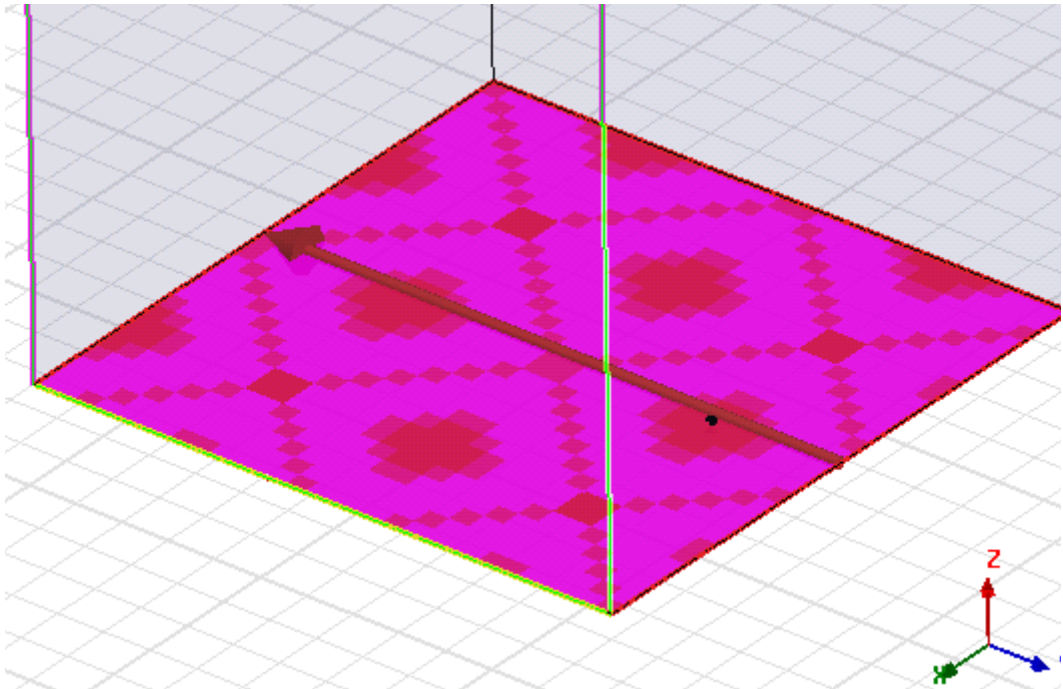


## Assign the Wave Port

To assign the wave port:

- 1** Select the bottom face of the lower box.
- 2** Right-click, and from the shortcut menu, select **Assign>Excitations>WavePort**.  
The WavePort wizard appears.
- 3** On the General page, accept the name Waveport1 and click **Next**.
- 4** See that the number of Modes is 2.
- 5** For the Mode one Integration line, select New line, and draw the line as shown, moving from the Y side towards

the X axis.

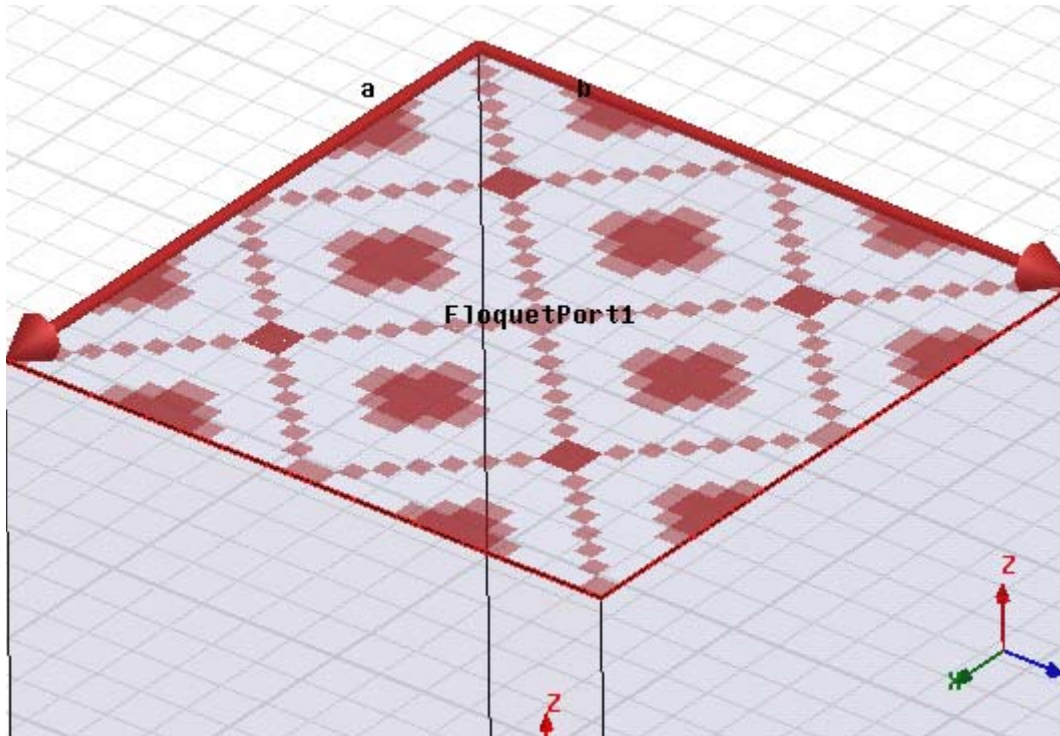


- 6** Accept the other settings, using Next to go through the other pages, and click OK.  
The waveport appears in the Excitation list in the Project tree.

## Assign the Floquet Port

To assign the Floquet port:

- 1 Select the top face of the upper box.
- 2 Right click, and select **Assign>Excitation>Floquet Port** from the shortcut menu.  
The Floquet wizard appears.
- 3 For the Lattice Coordinate System, Assign the A Direction and B Direction as shown.



- 4 Click Next, accepting the Phase Delay defaults, and Next again to the Modes Setup page.  
The default mode table specifies a pair of Floquet modes. The default modes both have modal indices  $m$  and  $n$  equal to zero and are sometimes called the "specular" modes. In the case at hand, 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, but sometimes on



the basis of electrical symmetry, one of the two polarizations may be omitted.

The final column of the mode table is labeled "Attenuation". This is the attenuation of the mode along the direction normal to the plane of the array in units of dB per model length unit. The value given for the specular modes is 0 dB since they are propagating and are therefore not attenuated. Higher-order modes will experience attenuation if they are cut off. Modes that are cut off at a certain scan angle may become propagating as the scan angle is increased.

**5** Set the Number of Modes as 2.

This fills out the Modes table with four entries. In general, 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.

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

**6** See that the Frequency is set to 299.79 Mhz, and select the radio button Use Scan Angles to Calculate Lattice Phase Delays.

Almost always you should select the top radio button to use Scan Angles on this panel and on the corresponding panel in the Slave boundaries. This automatically synchronizes the phase of the Floquet port with that of the linked boundaries, which is essential for a correct solution. The scan angles in were filled in automatically when the scan angles were previously set in the Slave boundary setup panels.

**7** See that the Scan Angles for Theta and Phi are 90 degrees.

**8** Click OK to close the dialog.

The 3D Refinement tab controls which Floquet modes par-

ticipate in 3D adaptive mesh generation. The default setting the Floquet modes will not participate in 3D adaptive refinement.

To understand what this means, recall that 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 principal interest. Hence the default setting of the 3D refinement tab which ignores the Floquet modal field patterns in adapting the 3D mesh.

The Post Processing tab specifies 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.

- 9** Accept the other default settings, and OK to close the Floquet wizard.

The Floquet port appears in the Project tree under Excitations.

## Create the Project Variables

To create project variables:

- 1** On the menu bar, click **Project>Project Variables**.  
This displays the Properties window for the Project.
- 2** Click the **Add** button.  
This displays the Add Property dialog.

- 3** Set the Name as  $\phi_{\text{scan}}$ , and the value as 0 deg.
- 4** Click OK.  
This closes the **Add Property** dialog adds  $\phi_{\text{scan}}$  as a variable in the project **Properties** window.
- 5** Click the **Add** button to display the **Add Property** dialog again.
- 6** Set the name as  $\theta_{\text{scan}}$  and the value as 0 deg.
- 7** Click OK.  
This closes the **Add Property** dialog adds  $\theta_{\text{scan}}$  as a variable in the project **Properties** window.
- 8** Click OK to close the project **Properties** window.

Also, it should be mentioned, 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 required, adhering to this natural convention insures the simplest and clearest relationship between scan angles and the structure being modeled.

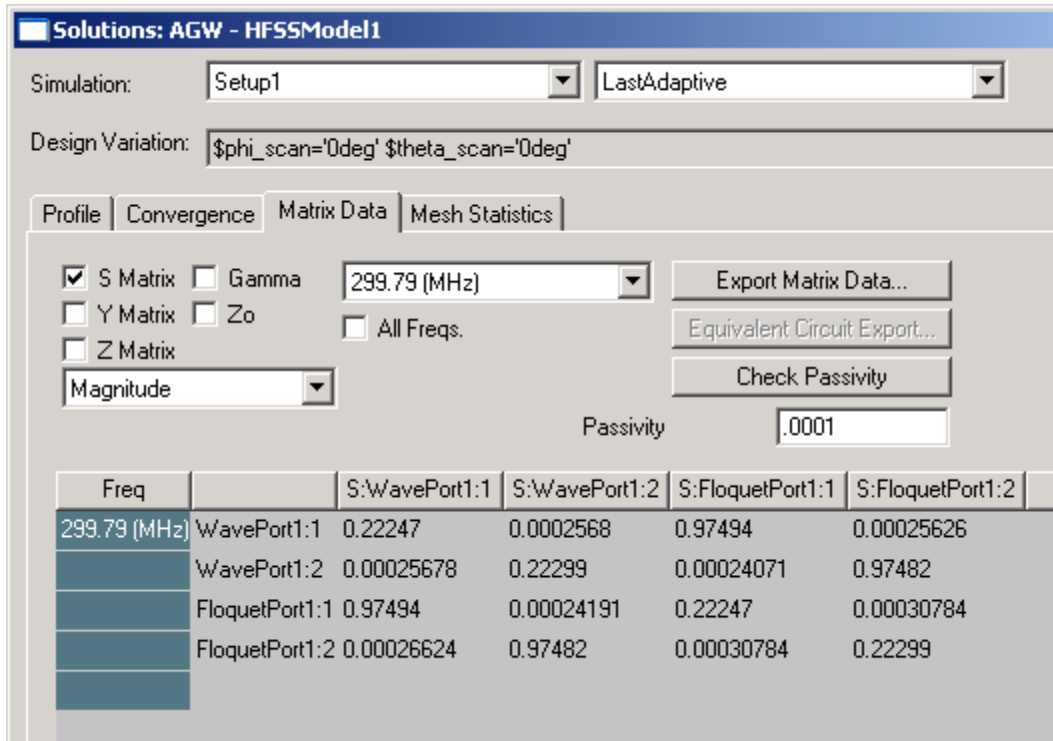
## Setup the Analysis

To setup this analysis:

- 1** Right click on Analysis in the Project tree, and select Add Solution Setup.  
This opens the Solution Setup dialog.
- 2** On the General tab, set the Solution frequency to 299.79 Mhz. Set the Maximum Number of Passes to 5, and the Maximum Delta S as 0.02.
- 3** On the Options tab, check the Do Lambda Refinement box, and set the Lambda target Use Default.
- 4** Set the Maximum Refinement Per Pass to 30%, the Minimum Number of Passes to 5, and the Minimum Converged Passes to 2.
- 5** Select First Order Basis function, and click OK to accept the Setup.

## Run Analysis and View Results

Use HFSS>Analyze to run the analysis. After completion, right click on the Results icon in the Project tree, and select Solution Data.



The figure shows matrix data panel after the simulation is complete. Since phase is unimportant for this case, only magnitudes are shown. There are several things to note.

- The S-matrix is a 4×4 matrix and inter-relates 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 TE<sub>00</sub> Floquet mode and FloquetPort1:2 refers to the TM<sub>00</sub> 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, 0.222, is the mag-

nitude 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".

## 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 beautiful 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 on Optimetrics in the Project tree, and select **Add Parametric**.

This displays the **Setup Sweep Analysis** dialog with the Sweep Definitions tab selected.

- 2** Click the **Add** button.

This displays the **Add/Edit Sweep** dialog.

- 3** From the Variable drop down menu, select  $\theta_{\text{scan}}$ .

- 4** Select Linear step.

- 5** Set the Start as 0 deg, the Stop at 90 deg, and the Step as 3 deg.

- 6** Click the **Add** button to accept the settings, and OK to close the **Add/Edit Sweep** dialog.

The Setup Sweep Analysis dialog includes the  $\theta_{\text{scan}}$

variable definition.

- 7** Click the General tab and see that the Sim Setup has Setup1 with the Include checked.
- 8** Click the Options tab and see that Save Fields and Mesh is not checked, and that Copy Geometrically Equivalent Meshes is checked.
- 9** Click OK to close the dialog.  
The Parametric setup is listed under Optimetrics.

## Prepare the Modes for the Parametric Sweep

- 1** Reenter the Floquet port setup panel and click on the modes tab.

The current pair of modes, appropriate for broadside scan, will need to be supplemented by additional modes for larger scan angles. We will use the modes calculator to determine a set of modes adequate for all scan angles of the simulation.

- 2** Invoke the modes calculator by clicking on the modes calculator button.

These inputs, in addition to the known lattice vectors, are the additional information required to create a set of recommended modes for the Floquet port. They are purely for the purpose of the mode selection and do not affect the model set up. Therefore we may freely vary the choices here to explore different scenarios and the effect on the recommended set of modes.

With this in mind, we will do the following:

- 3** Select 10 modes for the port. Most likely we will trim the resulting list, but requesting this many gives a good picture of what modes are necessary and what are not.
- 4** Set the frequency to 299.97 MHz, which is the actual frequency of simulation. 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.

- 5** Select to choose scan angles rather than phase angles and set the phi scan angle to 90 degrees (E-plane) and the theta scan angle to the largest value that will be used, here equal to 90 degrees.

The most extreme scan angle should require the largest number of propagating modes of all scan angles, so we select the Floquet modes to use on this basis.

- 6** Click OK to leave the calculator and to compute and view the recommended list of modes. The modes table is as appears in the Figure.

General | Phase Delays | **Modes Setup** | 3D Refinement | Post Processing

Number of Modes:

(Please click on this button if you want assistance in setting up modes.)

	Mode	Polarization State	m	n	Attenuation (db/length)
	1	TE	0	-1	0.00
	2	TM	0	-1	0.00
	3	TE	0	0	0.00
	4	TM	0	0	0.00
	5	TE	-1	-1	65.60
	6	TE	1	-1	65.60
	7	TM	1	-1	65.60
	8	TM	-1	-1	65.60
	9	TE	1	0	81.10
	10	TE	-1	0	81.10

We note the following.

- There is a new pair of propagating modes, the TE<sub>0</sub>-1 and TM<sub>0</sub>-1. The propagating nature of these modes, at the

scan angle of 90 degrees used to select the list, is indicated by the attenuation value of 0 dB.

- The specular modes  $TE_{00}$  and  $TM_{00}$  are still in the list, numbered as modes 3 and 4.
- There are now 6 additional modes in the list, numbered 5 through 10, all cut off, with the attenuation levels given in the right-most column.

At this point we can accept this list by clicking OK and run the simulation. However, it is always good policy in terms of simulation efficiency as well as ease of interpretation of results to eliminate unnecessary modes.

Since modes 5 through 10 are cut off, even at the extreme scan angle, they are candidates for exclusion. The first pair, modes 5 and 6, have an attenuation of 64.97 dB per meter. Since the length of the unit cell is 1.25 meters, either of these modes generated at the rectangular aperture will experience attenuation of  $1.25 * 64.97 = 81.21$  dB by the time it reaches the Floquet port. This suggests that these modes may be neglected in the simulation. A similar picture applies to modes 7 through 10.

- 7** Therefore eliminate the cutoff modes from consideration by decreasing the number of modes in the Number of Modes box to 4.

Number of Modes:

(Please click on this button if you want assistance in setting up modes.)

Modes Calculator...

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

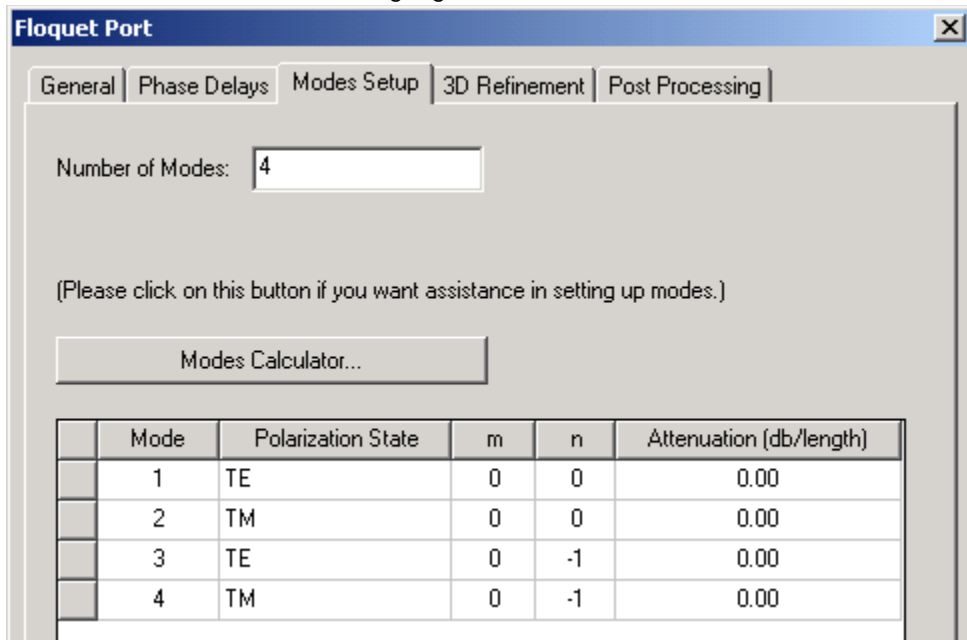


Note that the list is trimmed from the bottom up and now only the TE0-1, TM0-1, TE00 and TM00 remain.

- 8** As a matter of preference, we might like to have the specular modes listed first in the computed S-matrix.

Therefore move them to the top of the list by dragging each corresponding line by the square box at the left of each row.

The final appearance of the modes table is shown in the following figure.

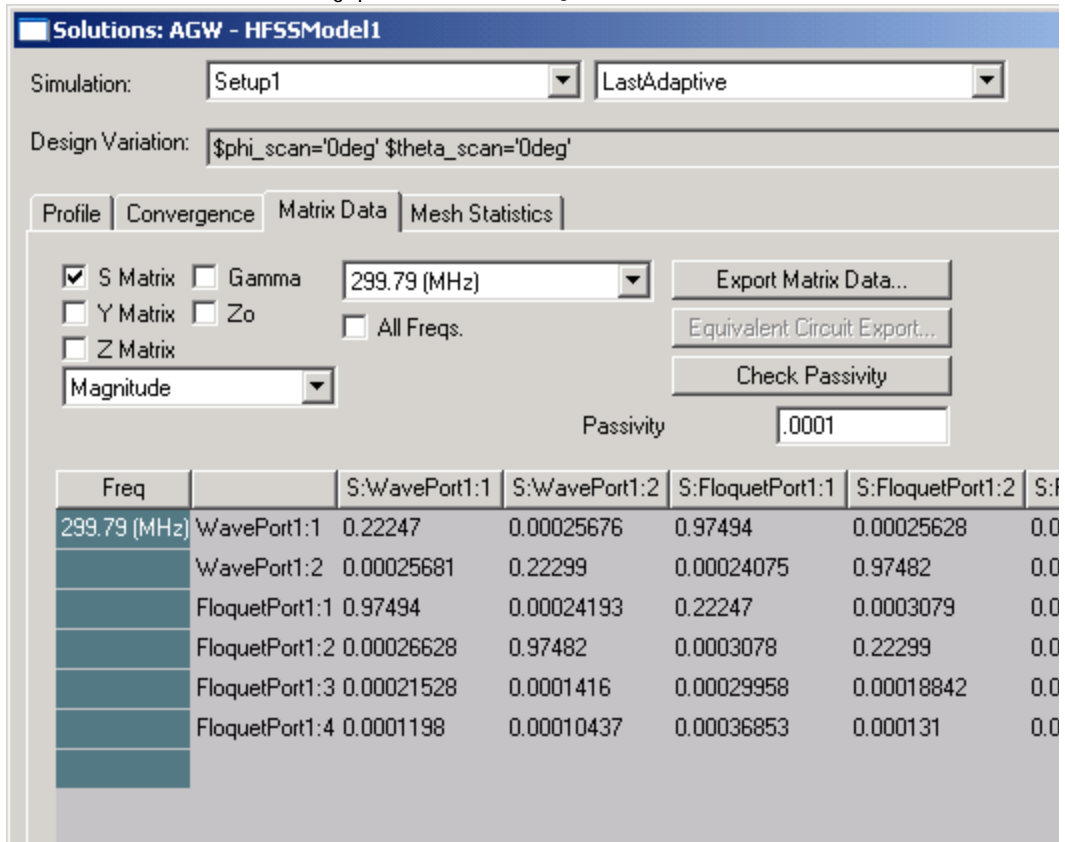


- 9** Click OK to accept this set of modes and then run the parametric sweep. (This will take some time.)

## Viewing the Results of the Parametric Sweep

Once the simulation is complete, the S-matrix elements as a function of scan angle can be examined in the Matrix Data panel or the Reporter. 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.



To create a report showing the (active) reflection and the TM transmission magnitudes plotted as a function of scan angle:

- 1 Right click on Results in the Project tree, and from the short cut menu select **Create Modal Solution Data Report>Rectangular Plot**.  
The **Report** dialog appears.
- 2 On the Trace tab, X: drop down menu, select  $\theta_{scan}$ .
- 3 For Y:, select S Parameter as the Category, S(Wave Port1:1) as the Quantity and Mag as the Function. This corresponds to the Reflection.
- 4 Click OK to create the new report.

The new report is displayed and added under Results in the Project tree with the first trace listed under the plot. The Add Trace button is enabled in the Reports dialog.

- 5** Select name of the trace in the Project Tree.

This displays the docked property window for the Trace

- 6** Check the box for Specify Name to enable the Name field, and change the text to Reflection.

This changes the displayed name of the first trace in the Legend in the Report.

- 7** Add two additional traces and changes their names:

$\text{mag}(S(\text{WavePort1:1}, \text{FloquetPort1:2}))$  corresponding to  $\text{TM}_{00}$  Transmission

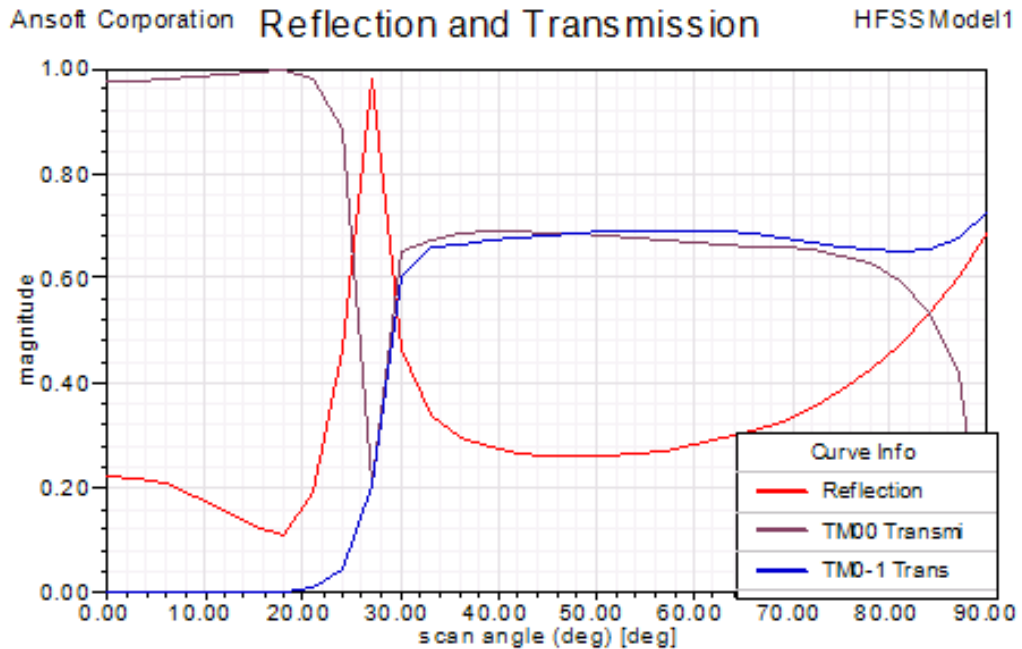
$\text{mag}(S(\text{WavePort1:1}, \text{FloquetPort1:4}))$  corresponding to  $\text{TM}_{0-1}$  Transmission

- 8** In the Project tree select X Y Plot 1.

This displays the Properties for the plot in the docked properties window.

- 9** Edit the Name field to Reflection and Transmission and press enter.

The plot should resemble the following figure.



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 guides you through the creation and simulation of the frequency selective surface model.

- ✓ Create the Unit Cell
- ✓ Assign the Boundaries
- ✓ Assign the Floquet Ports
- ✓ Setup the Analysis
- ✓ Run the Simulation
- ✓ Create the Reports

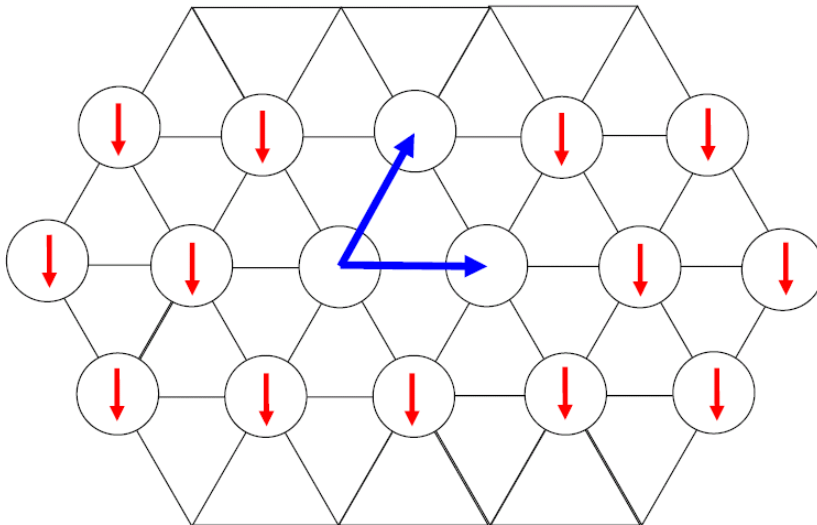
## Frequency Selective Surface Example

Unit cells for frequency selective surface (FSS) simulations may be constructed using linked boundaries and two Floquet ports, with one port above the plane of the structure and one port below. 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.

### Example Model

The example to be considered here is a conducting screen containing a rhombic lattice of circular apertures. The lattice geometry is shown in Figure 1 in which a pair of lattice vectors are drawn in blue. The angle between the lattice vectors is 60 degrees.



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.

A simple unit cell model has been created and is shown in Figure 2. 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.

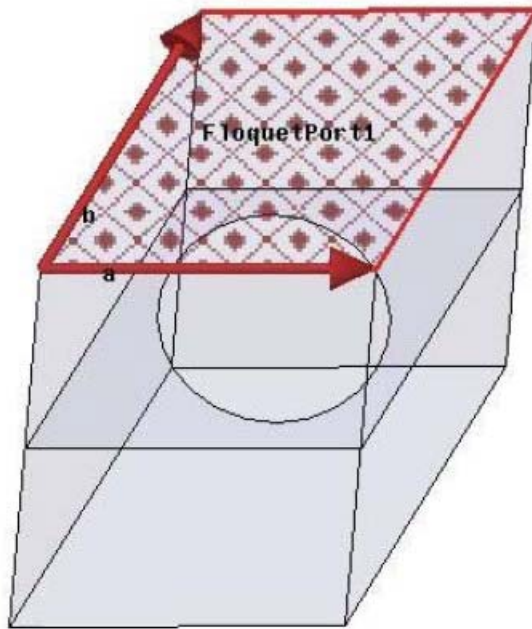


Figure 2- Unit cell for rhombic lattice

## Create the Unit Cell Model

A simple unit cell model has been created and is shown in Figure 2. 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.

## Create the Rhombic Object

To create the rhombic object.

- 1** Open a new project and name it rhombicArray
- 2** Click **Draw>Line**, click an arbitrary point to set the starting point, click three more arbitrary points, then return to the close the line by clicking again on the starting point.
- 3** Right click to display the shortcut menu and click Done.  
This creates an initial the polyline object.
- 4** Look in the History Tree and expand it under **Sheets>Unassigned>CreatePolyline**, select the first CreateLine to display the docked properties for that line.
- 5** Edit the Point1 coordinates to 0,0,0 in cm units.
- 6** Edit the Point2 Coordinates to 0.865, 1.4982, 0 in cm units.  
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 values as follows:

CreatLine	Second	Third	Fourth
Point1	0.856, 1.4982, 0	2.595, 1.4982, 0	1.73, 0, 0
Point2	2.595, 1.4982, 0	1.73, 0, 0	0, 0, 0

This creates the rhombic sheet that you can use to create the rhombic cell. You may wish to make a copy of the object as a first step in creating the aperture object.

- 8** To transform the 2D sheet object into the 3D cell object select the it and click **Draw>Sweep>Along Vector**.  
This displays the Measure Data dialogue, and also enables the entry fields on the Status bar for coordinates and



units. The status bar message says "Input the first point of the sweep vector."

You can enter the sweep vector either by clicking or via the status bar. In this example, use the status bar.

- 9** On the status bar, enter 0 in the X, Y, and Z cells to define the starting point, tabbing to change cells. Accept the defaults for the coordinates and units and press Enter.

This places an indicator on the first vector point, and changes the status bar message to "Input the second point of the sweep vector." The cell labels show dX, dY, and dZ, for the distance.

- 10** In the status bar, enter 0, 0, and 3, accepting the default coordinates and units, and press Enter.

This draws an indication line from the starting point to the end point, and displays the Sweep Along Vector dialog.

- 11** In the Sweep Along Vector dialog, accept 0 deg as the draft angle, and select round from the drop down menu as the Draft type. Then click OK.

- 12** This completes the sweep., converting the sheet object into a solid.

- 13** Next move the Object by selecting the object and clicking the Move icon.

- 14** This opens the Measure dialog and the enables the X, Y, and Z cells on the status bar for you to Input the reference point for the move vector.

- 15** On the status bar, set the Z value to 0, and press Enter.

- 16** In the status bar, set the dZ value to -1.5 and press Enter.

This moves the rhomboid to the proper position.

## Create the Aperature Object

If you have made a copy of the rhombic sheet, you can use that. If not, repeat the process in steps 1- 7.

- 1** Click the copy of the rhombic sheet.
- 2** Select the properties and specify the X and Y dimensions as 1.73 cm.

- 3** Click **Draw>Circle** draw circle inside the rectangle by holding the cursor over the center of the rectangle, where a round dot will appear under the cursor. When the dot appears, click the button, and drag towards the edge of the rectangle and click to draw the circle.
- 4** Open the properties dialog for the circle and set the radius as 0.6 cm.
- 5** Select both the circle and the rectangle, and click **Modeler>Boolean>Subtract**.  
This opens the **Subtract** dialogue.
- 6** Move the Rectangle name to the **Blank** parts list and the circle name to the **Tool** parts list.
- 7** Click **OK** to close the dialogue and create the aperture by subtracting the circle from the rectangle.

### **Move the Aperture to the Rhomboid**

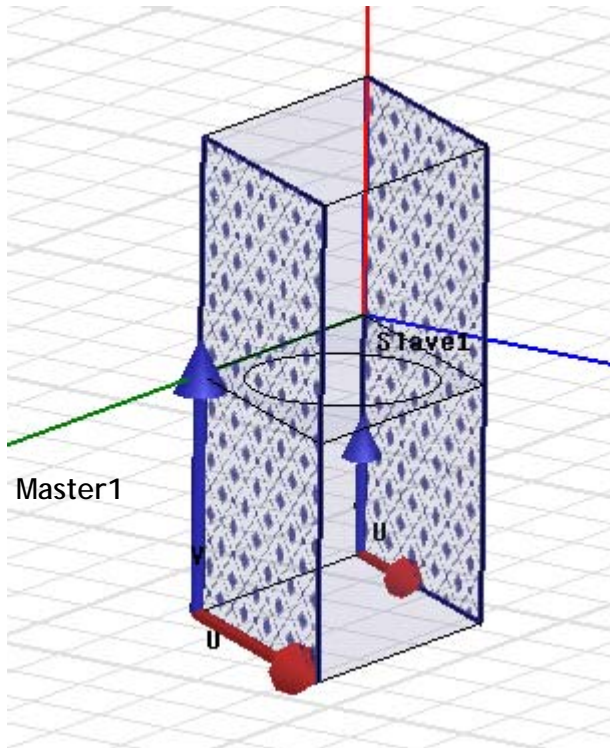
To move the aperture to the rhomboid object:

- 1** Select the aperture object and click the Move icon.
- 2** Drag the aperture object inside the rhomboid.

## Assign the Master and Slave Boundaries

Assign the Master and Slave boundaries to the rhomboid object as follows.

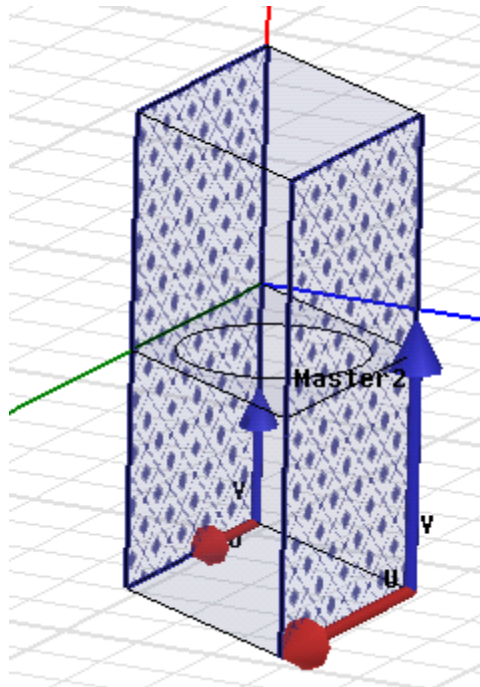
- 1 Select the face shown in the figure, and click **HFSS>Boundaries>Assign>Master...** or right click on Boundaries in the Project tree and select **Assign>Master...**



The Master Boundary dialog appears.

- 2 Accept the default name as Master1.
- 3 Click the drop down menu for the U vector, and click, New Vector.  
The Measure dialog and a the Create Line prompt appears.
- 4 Draw the U vector of the coordinate system on the selected face. Click on the lower left corner as the starting point, and drag the cursor to the right corner and click.

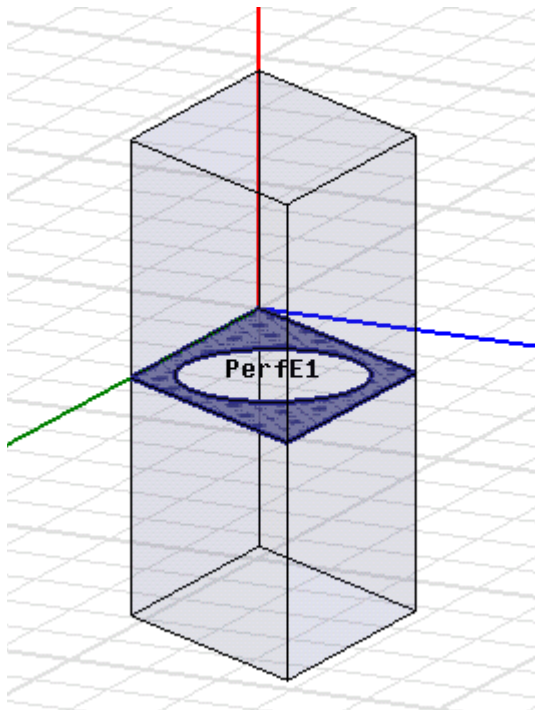
- 5 In the **Master Boundary** dialog, for the V Vector, check the box for Reverse direction, and OK to close the dialog.
- 6 Select the opposite face, and **HFSS>Boundaries>Assign Slave...** or right click on Boundaries in the Project tree and select **Assign>Slave...**  
The Slave dialog appears with the General Tab selected.
- 7 Select Master1 as the Master boundary.
- 8 Draw the U Vector as shown. Accept the other settings and OK to close the dialog.
- 9 Repeat the procedure for the Master2 and Slave 2 boundaries, as shown. The main difference is that for the Master2 boundary, you do not check the Reverse direction dialog for the V vector, but you check it for the Slave2 boundary.



## Assign the Perfect E Boundary

For this section, you assign a Perfect E boundary to the aperture.

- 1 Select the aperture from the history tree.
- 2 Select **HFSS>Boundaries>Assign>Perfect E** or right click on the Boundary icon in the Project tree and select **Assign>Perfect E** from the shortcut menu.
- 3 Accept the default name and click OK to close the dialog.
- 4 The PerfE1 boundary appears in the list under the Boundary icon.



## Assign the Floquet Ports

This section helps you assign the Floquet ports to the top and bottom faces of the model.

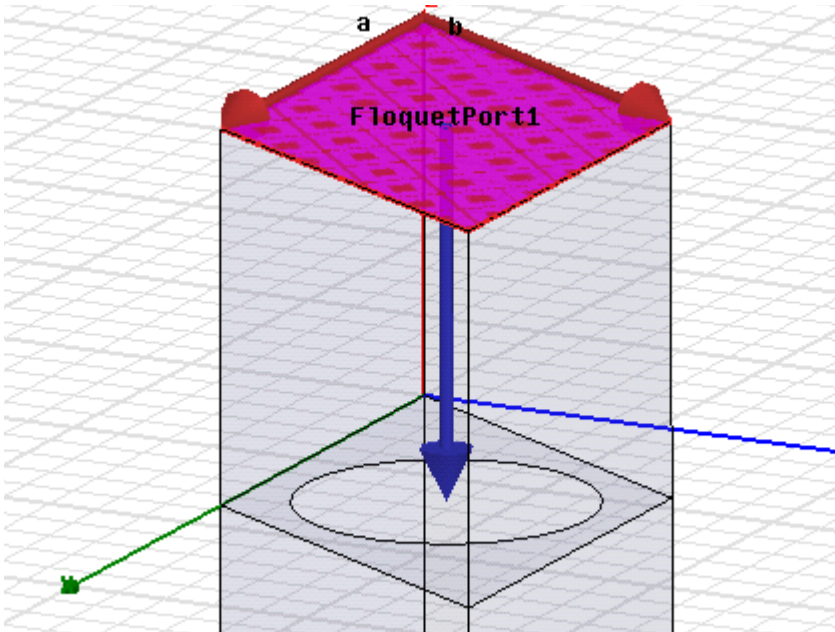
- 1 Select the top face of the model, and click **HFSS>Excita-**

tions>Assign>Floquet Port

The Floquet Port wizard appears, showing the General page.

- 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.



- 3** Draw the a vector as shown, clicking on the corner nearest the Z axis to start, and then clicking on the corner along the X direction.

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.

Repeat the process to draw the b vector as shown.

- 4** Click Next in the Floquet wizard to set the Phase Delays.
- 5** Select the Input Lattice Phase Delays radio button, and click Next to go to the Modes setup page.

- 6** On the Modes setup page, enter 14 as the Number of Nodes.  
This fills out the Mode table to 14 entries with initial 0 values.
- 7** Click the Modes Calculator button for assistance in setting up the modes.  
The Modes calculator appears.
- 8** Accept the values by clicking OK to close the Calculator.  
This fills out the table with calculated values.
- 9** Click Next for the 3D Refinement page.
- 10** For Mode 1, click the Affects Refinement checkbox, and click Next to for the Post Processing page.
- 11** Check the box for Deembed, and specify the Distance as 1.5 cm.
- 12** Click OK to close the dialog.  
The first Floquet port appears under the Excitations icon in the Project tree.
- 13** Select the bottom face of the model, and repeat the process to setup the second Floquet port.

## Setup the Analysis

To setup this analysis:

- 1** Right click on Analysis in the project tree, and select Add Solution Setup.  
This opens the Solution Setup dialog.
- 2** On the General tab, set the Solution frequency to 20 Ghz. Set the Maximum Number of Passes to 10, and the Maximum Delta S as 0.02.
- 3** On the Options tab, check the Do Lambda Refinement box, and set the Lambda target as 0.2.
- 4** Set the Maximum Refinement Per Pass to 20%, the Minimum Number of Passes to 6, and the Minimum Converged Passes to 2.
- 5** Select First Order Basis function, and click OK to accept the Setup.

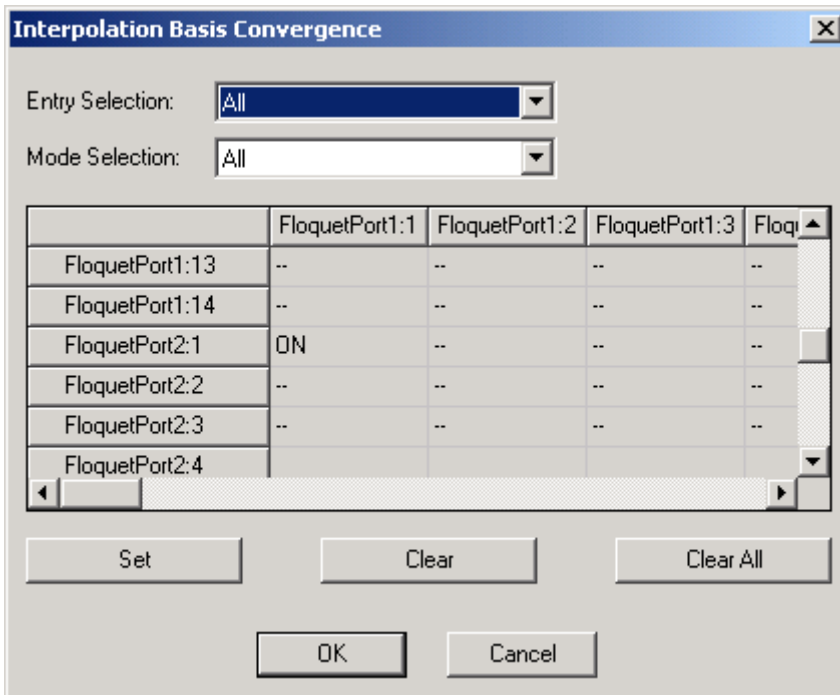
## Setup the Sweep

To setup the Sweep:

- 1** Rightclick on Setup1 in the Project tree, and select **Add Frequency Sweep**.  
The **Edit Sweep** dialog appears.
- 2** For the Sweep Type, select **Interpolating**.
- 3** For the **Frequency Setup**, select **Linear Step**. Then set the Start as 8 Ghz, the Stop as 20 Ghz, and the Step Size as 0.2 Ghz.
- 4** For the **Interpolating Sweep** options, set the Max Solutions as 50, and Error Tolerance at 0.5%.
- 5** Click the **Advanced Options** button to display the **Interpolating Sweep Advanced Options** dialog.
- 6** Set the Minimum Solutions to 5, and the Minimum number of Subranges to 1.
- 7** For **Interpolation Convergence**, select the **Use Selected Entries Radio** button, and Click the **Select Entries Button**.  
The **Interpolation Basis Convergence** appears.
- 8** In the **Interpolation Basis Convergence** dialog, leave the **Entry and Mode Selections** as **All**. Then use the vertical scroll bar to find the **FloquetPort2:1 Row**, and set the



value in the FloquetPort1:1 column on that row to ON.

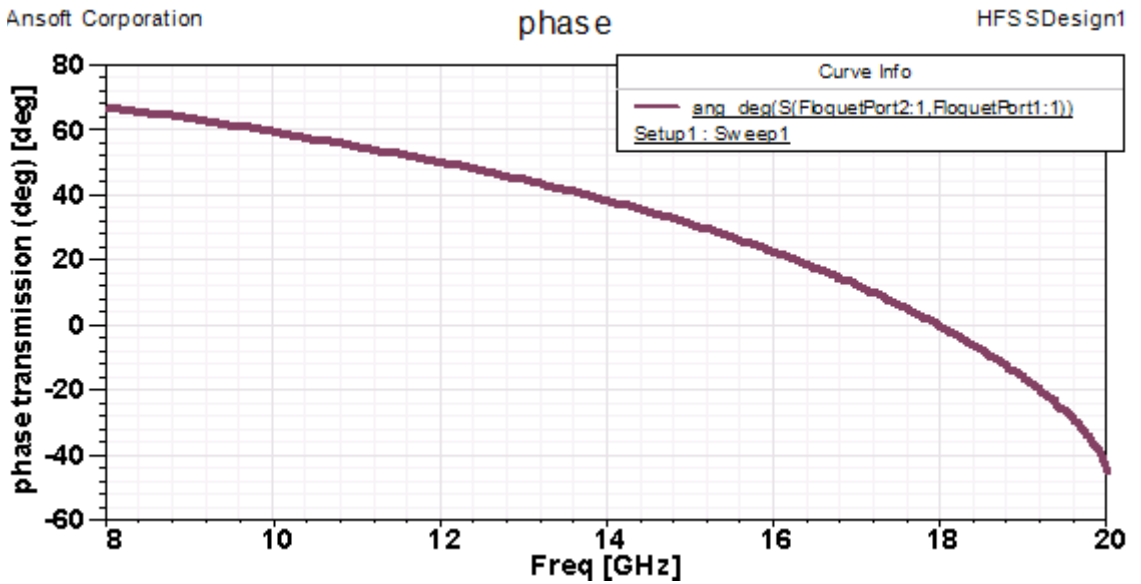


- 9 Use the Horizontal and Vertical scroll bars to find the row for FloquetPort1:1 and the column for Floquet Port2:1, and set that cell to ON.
- 10 Then OK to close the dialog, OK to close the Interpolating Sweep Advanced Options, and OK to close the Edit Sweep dialog.
- 11 Right click on the Sweep in the Project tree and click Analyze on the shortcut menu to start the simulation.  
This will take some time.

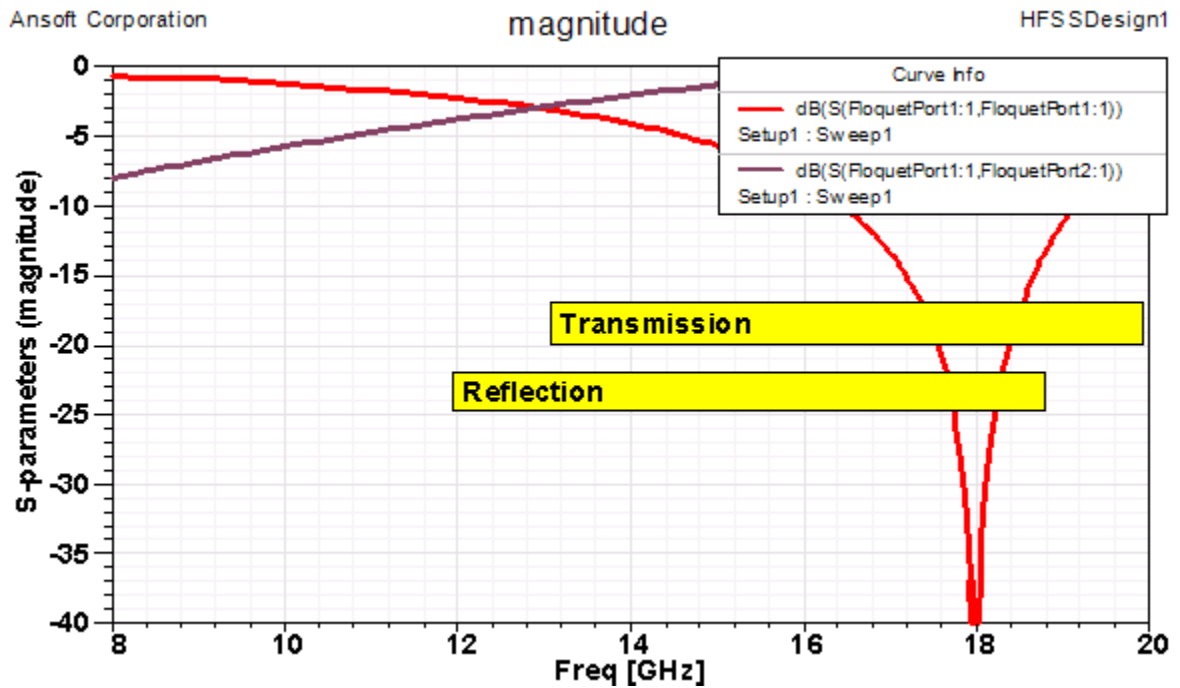
## Viewing the Results

Reports on the results.

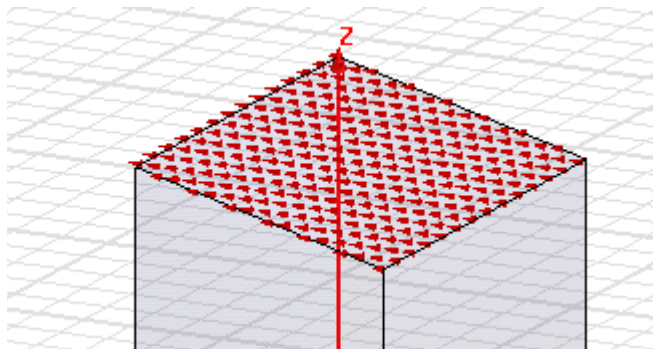
A report of the Phase.



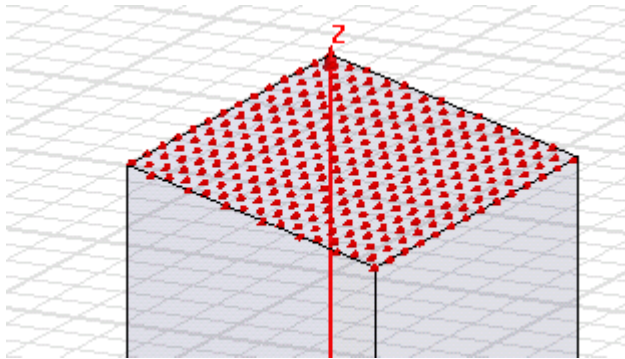
## A Report of the Magnitude



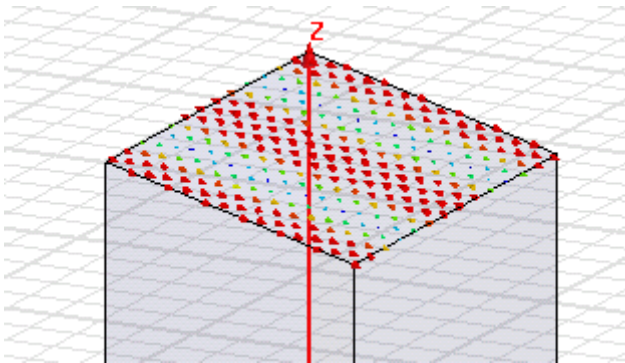
Some examples of the Port Field Displays for Modes.



Mode 2 Port Field Display



Mode 3 Port Field Display



Mode 4 Port Field Display

