Ansoft High Frequency Structure Simulator

Tutorial

TE_{10} Mode Cavity Resonator

V1.0

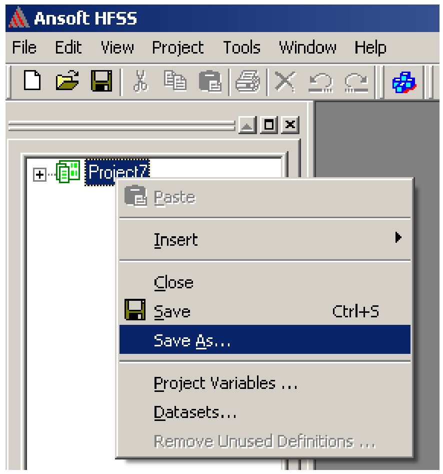
March 2004

Introduction

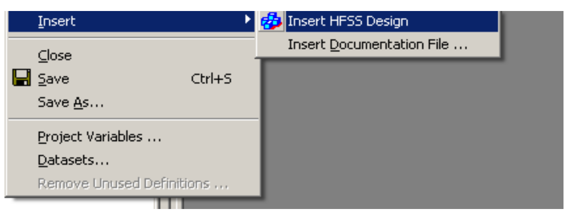
In this tutorial, a single-mode cavity resonator will be constructed and analyzed using the HFSS simulation software. This tutorial is designed for first time users and is intended to introduce commonly used commands and techniques. By the end of this tutorial the user should be able to:

- ✓ General navigation of software menus, toolbars, and quick keys.
- ✓ Variable assignment.
- ✓ Overview of commands used to create complex geometric models.
- ✓ Proper assignment of sources and boundaries.
- ✓ Analysis Setup.
- ✓ Report creation and options.
- ✓ Optimetrics setup and analysis.

From the Project Manager window, right-click the project file and select Save As from the submenu. Name the file “TE10_cavity” and Click Save.



To begin working with geometries, you must insert an HFSS design. Right-Click the project file and select Insert -> Insert HFSS Design from the menu.



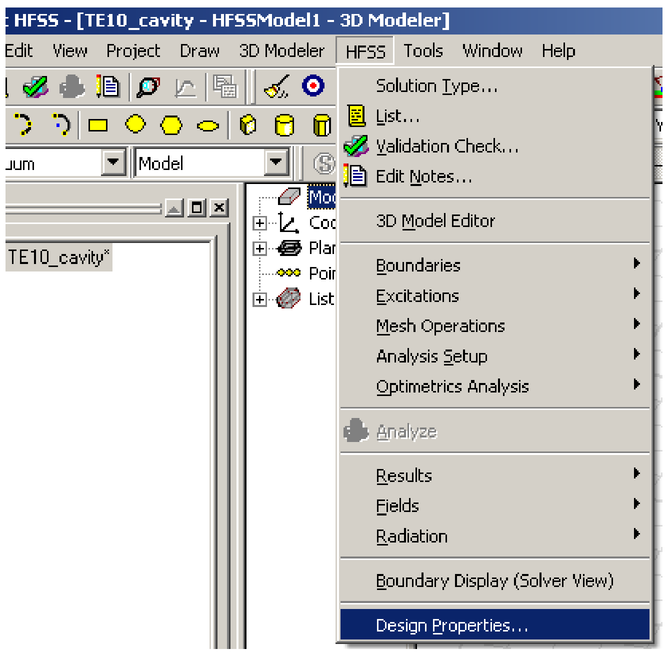
Always create a personal folder to store all HFSS projects. You may find you do not have access rights to some portions of the hard drive. This will also allow the user to quickly backup/copy data from projects.

- Note:

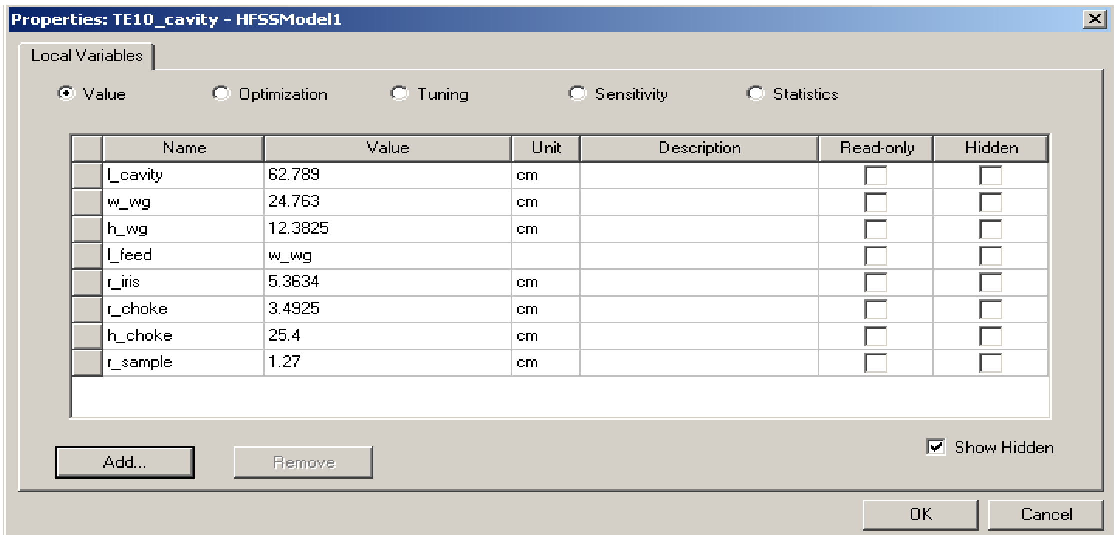
HFSS relies on variables for parameterization/optimization within the project. Variables also hold the following benefits:

- ✓ Fixed Ratios (length, width, height) are easily maintained using variables.
- ✓ Optimetrics use variables to optimize the design according to user-defined criteria.
- ✓ All dimensions can be quickly changed in one window as opposed to altering each object individually.

Click the HFSS heading and select Design Properties at the bottom of the menu.



A table will be displayed allowing the user to add the variables of the project. Click Add to enter the variables. Be sure to enter units immediately following the value with no space between. Enter all variables shown below:



Name	Value	Unit	Description	Read-only	Hidden
l_cavity	62.789	cm			
w_wg	24.763	cm			
h_wg	12.3825	cm			
l_feed	w_wg				
r_iris	5.3634	cm			
r_choke	3.4925	cm			
h_choke	25.4	cm			
r_sample	1.27	cm			

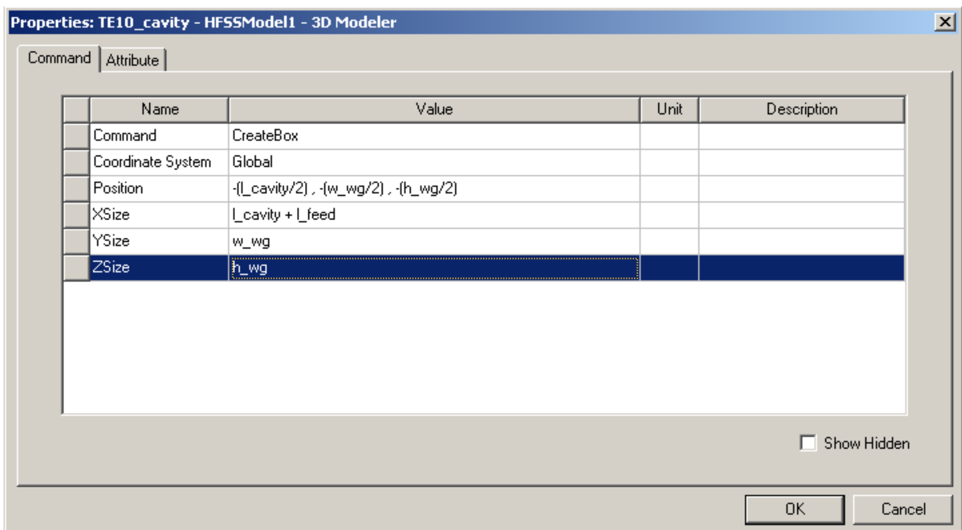
Click OK to complete.

➢ **Note:**

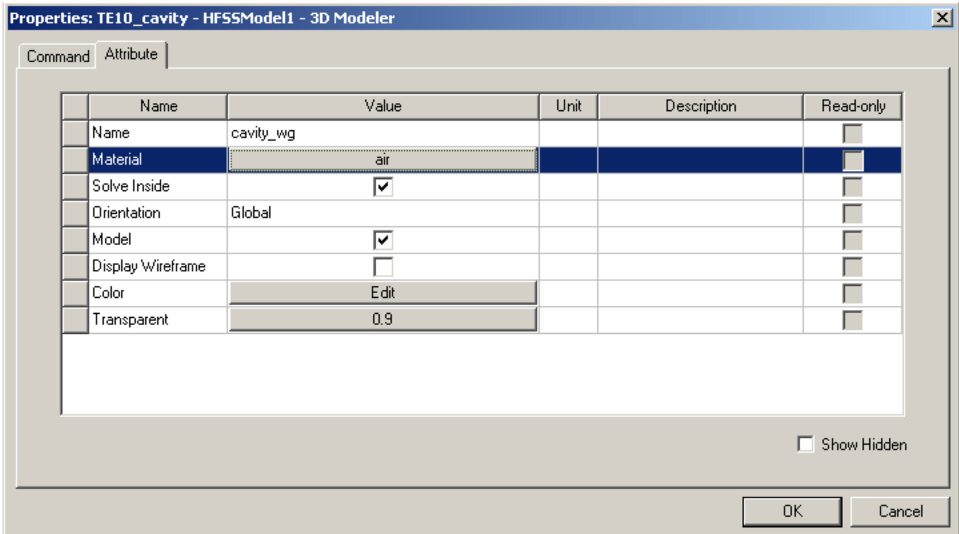
> Creating variables before defining the structure will allow the user to build the geometry much faster than using a fixed coordinate system.

With all variables defined, the model can be quickly constructed. All geometries in this tutorial are created using the 3D modeler toolbar. The toolbar is illustrated below:

We will begin by constructing the primary structure. This consists of a waveguide feed and a resonant cavity. Select Draw Box (from the toolbar) and use the mouse to select 3 arbitrary points within the drawing area. A box will display properties that will be entered as shown below:

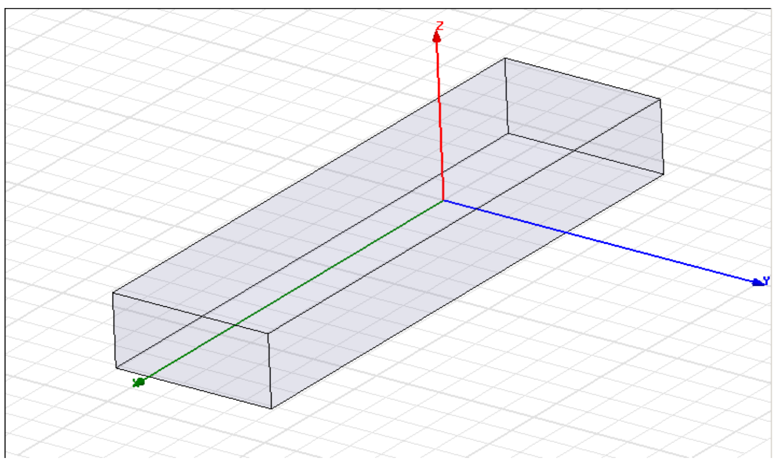


Under the Attribute tab and enter the following:

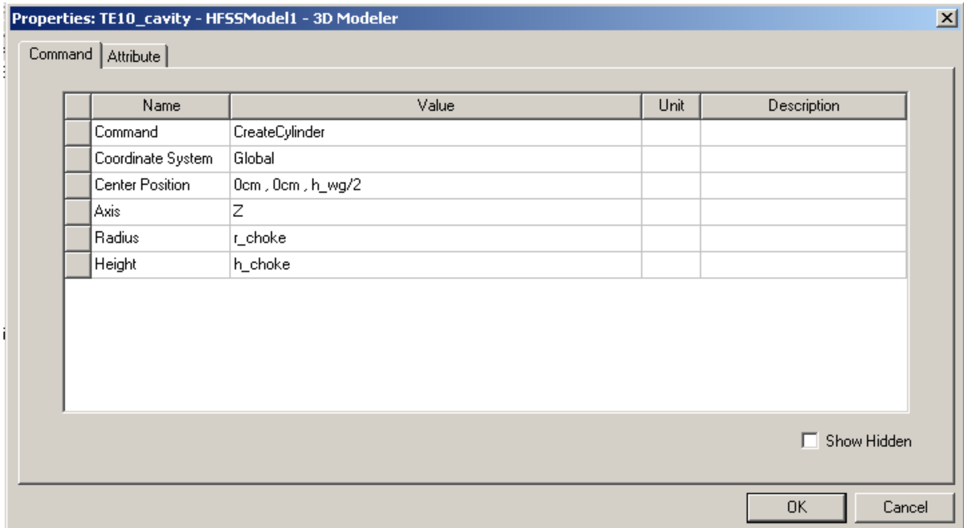


Building the Model

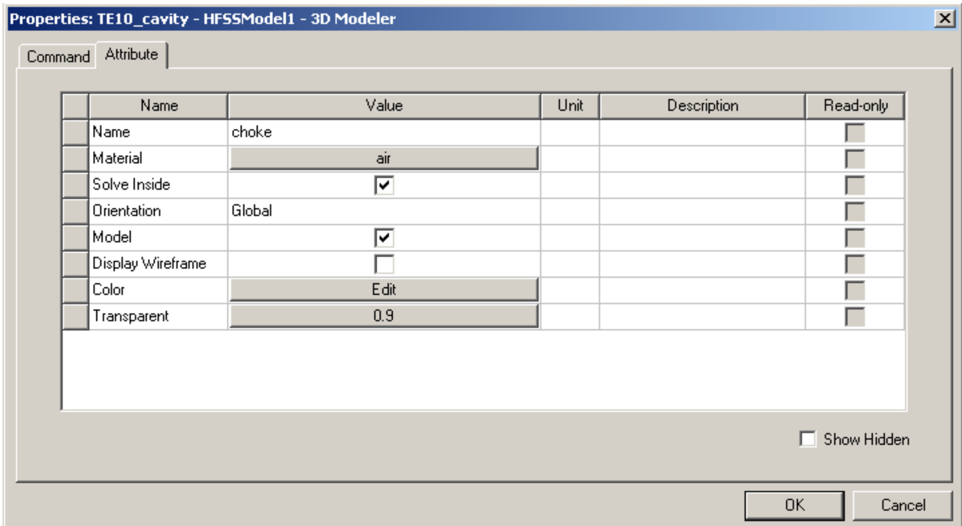
Click OK when complete. The structure is depicted below:



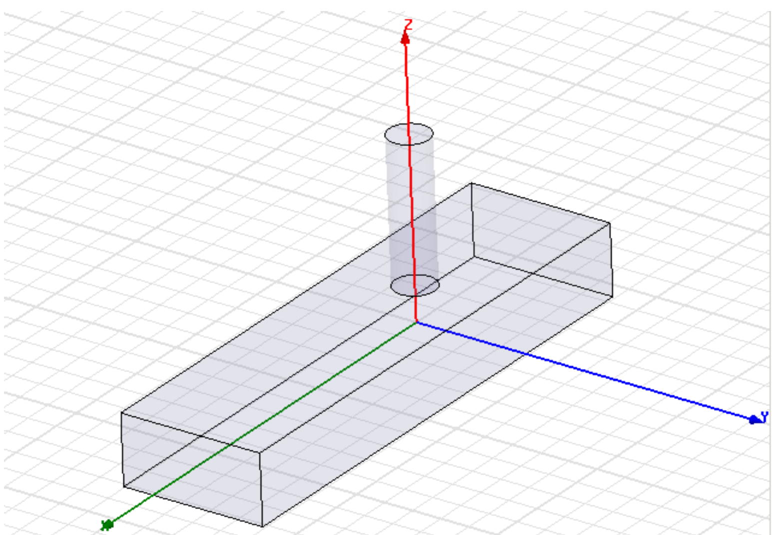
Next, we will build the "choke". We will make use of the duplicate command to construct it in two steps. Select Draw Cylinder and arbitrarily position the object in the drawing window. Enter the following properties:


Properties: TE10_cavity - HFSSModel1 - 3D Modeler
Command
Attribute
Name
Value
Unit
Description
Command
CreateCylinder
Coordinate System
Global
Center Position
0cm , 0cm , h_wg/2
Axis
Z
Radius
r_choke
Height
h_choke
Show Hidden
OK
Cancel

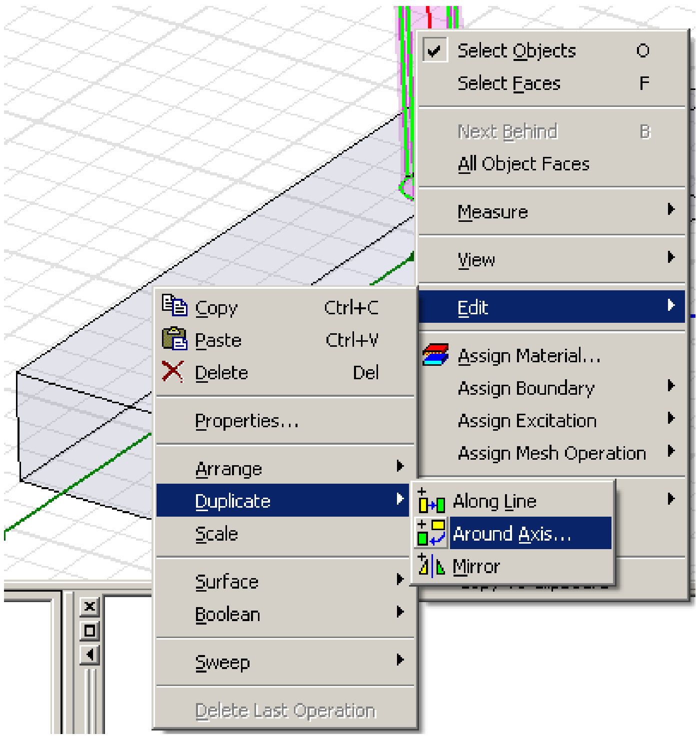
Under the Attribute tab, enter the following:



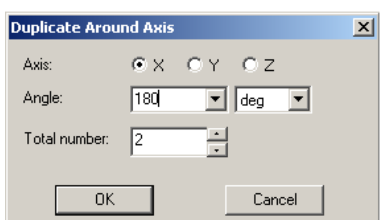
Click OK when complete. Step one of the “choke” is complete and should appear as follows:



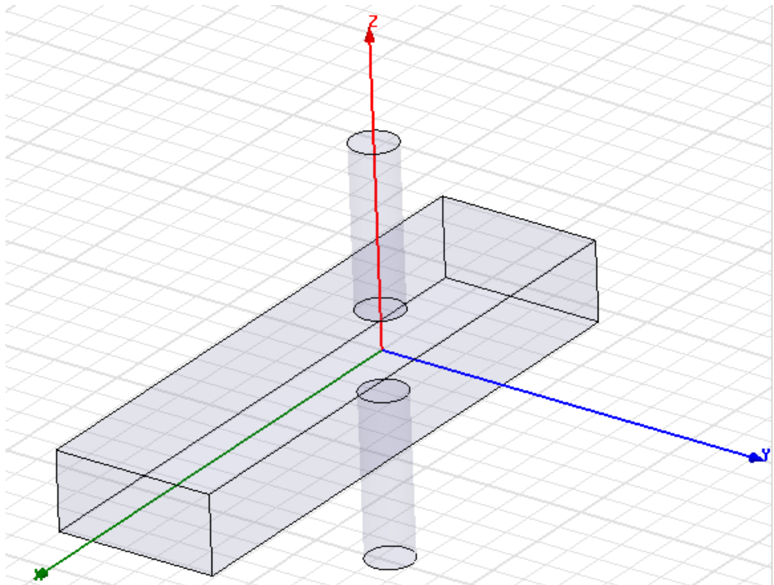
The next command is very useful when building symmetrical objects. Select the newly created "choke" by clicking the object directly. Right-click the modeling area and select Edit -> Duplicate -> Around Axis.



In the property window, enter the following:



The result of the Duplicate operation is shown below:

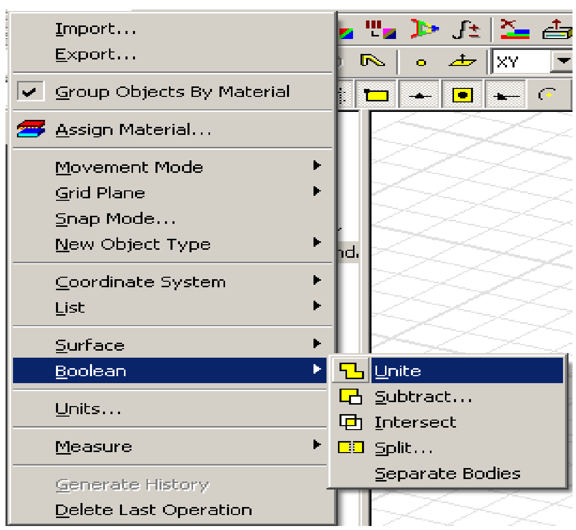


➔ Note:

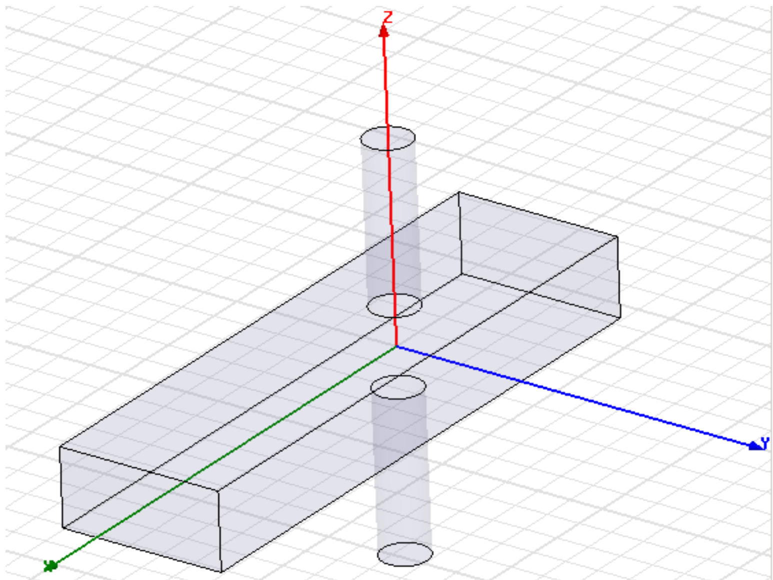
Make sure that all materials, up to this point,have been defined as air.

At this point, all three objects are considered as separate entities. Using the selection button on the toolbar, select all 3 objects OK.

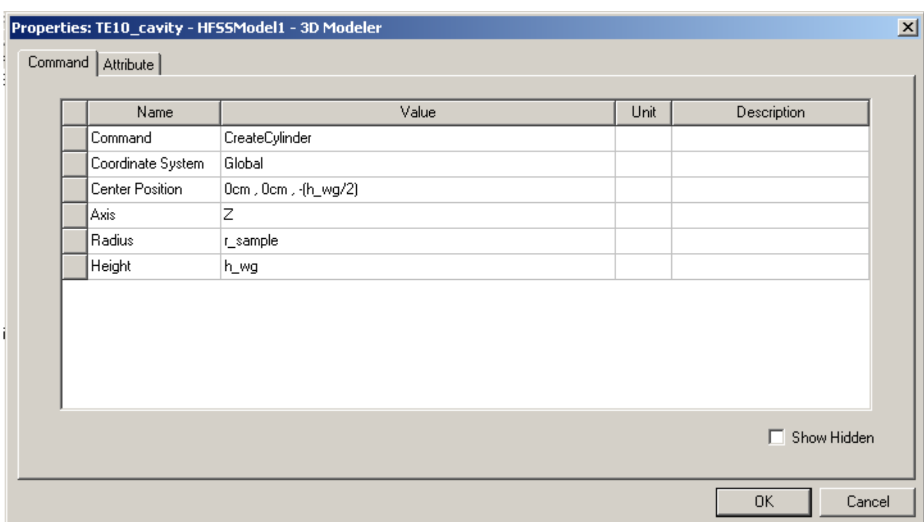
From the toolbar, select 3D Modeler -> Boolean -> Unite.



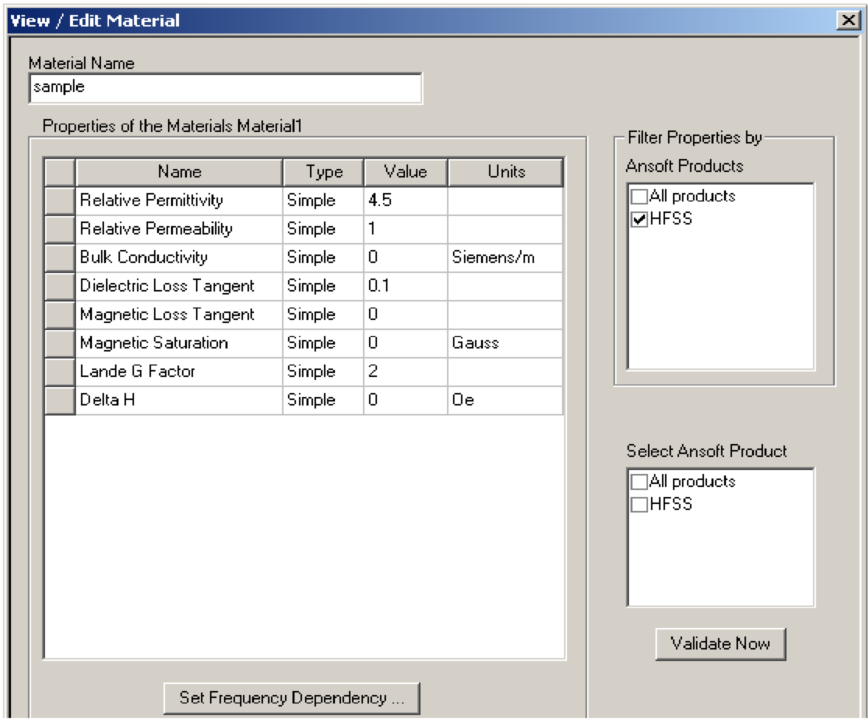
All three objects have now been united. This operation unites the "choke" structure with the "cavity" making the volume common.



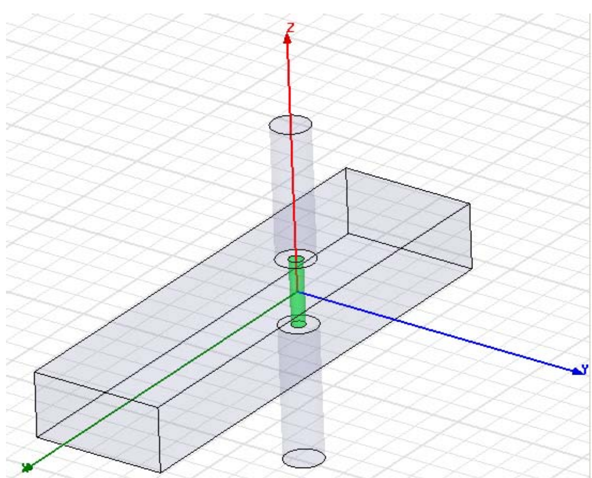
In the next step, the "sample" structure is created. A custom material, with a complex dielectric constant, will be defined. Begin by selecting Draw Cylinder from the 3D toolbar. From the property window, select the following:



Under the Attribute tab select Material and click Add Material enter “sample” for the name. Now create a custom material by selecting Material and enter the following:

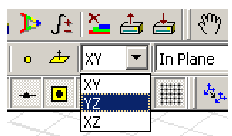


The material is defined as complex by defining the dielectric loss tangent. See help for further explanation of complex material assignment.

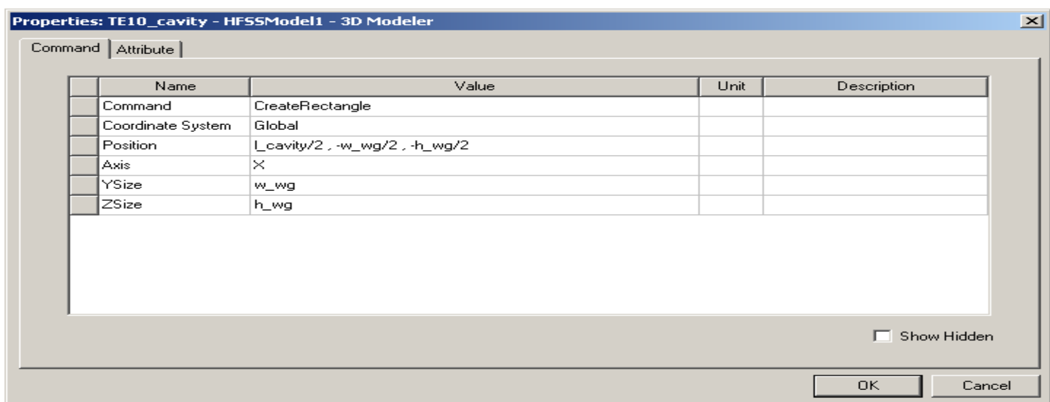


The final structure to add to the geometry is the “iris”. The iris allows microwave energy to be coupled into the cavity. The geometry and size of the iris affects the resonant characteristics of the cavity. This structure will be built in two steps. First, a plate is created, separating the waveguide and cavity. Next, the circular “iris” will be placed on the plate and subtracted. This operation will create a hole in the plate and the radius of this “iris” will be optimized for maximum energy transfer.

The two dimensional plate is placed in the yz-plane. Change the drawing orientation by selecting yz from the toolbar as shown below:

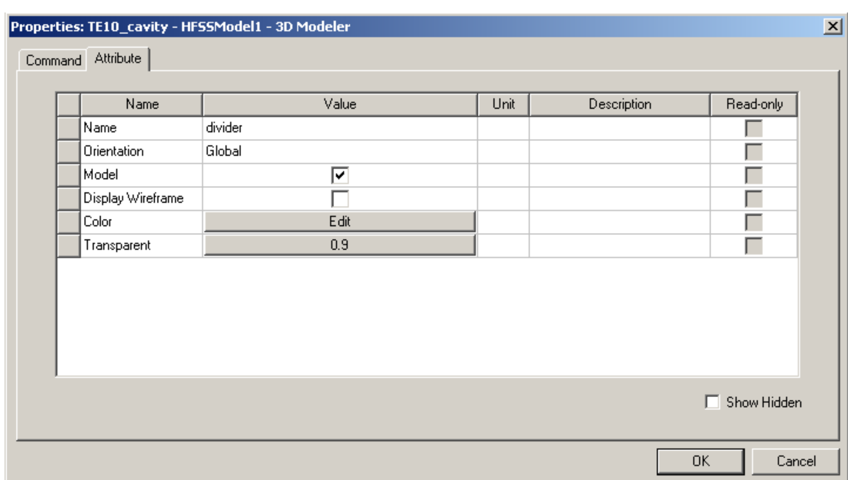


Select Draw Rectangle from the toolbar. Insert the following information:



Name	Value	Unit	Description
Command	CreateRectangle		
Coordinate System	Global		
Position	l_cavity/2 , -w_wg/2 , -h_wg/2		
Axis	X		
YSize	w_wg		
ZSize	h_wg		

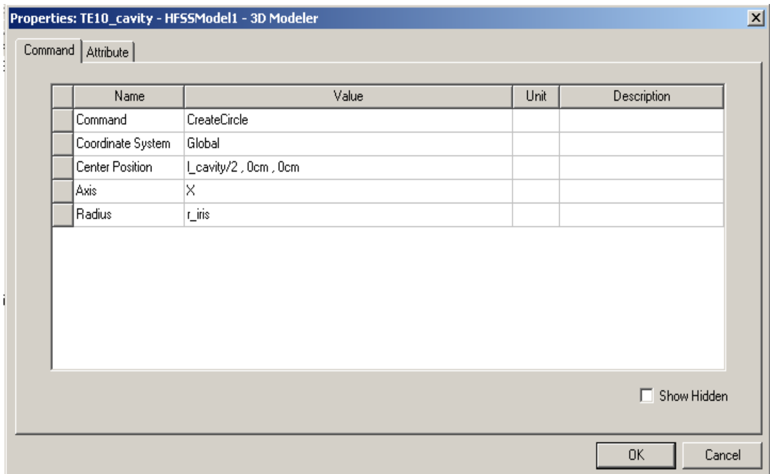
Select the Attribute tab and enter the following:



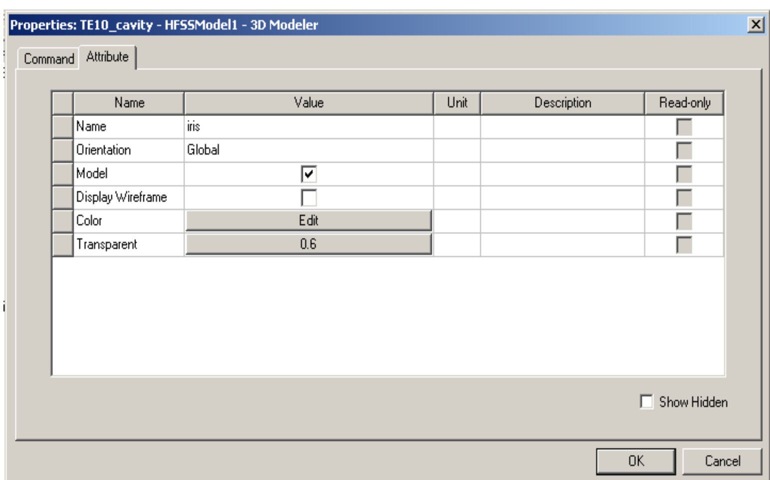
Name	Value	Unit	Description	Read-only
Name	divider			☐
Orientation	Global			☐
Model	☑			☐
Display Wireframe	☐			☐
Color	Edit			☐
Transparent	0.9			☐

Click OK when complete.

Since the plate is a two dimensional object, a material definition is not possible. Now create the “iris” by selecting Draw Circle from the toolbar and enter the following:

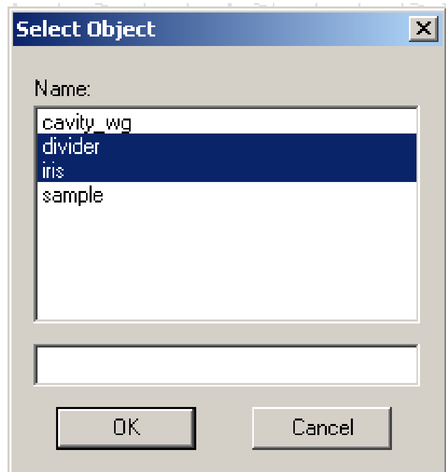


Under the Attribute tab, enter:

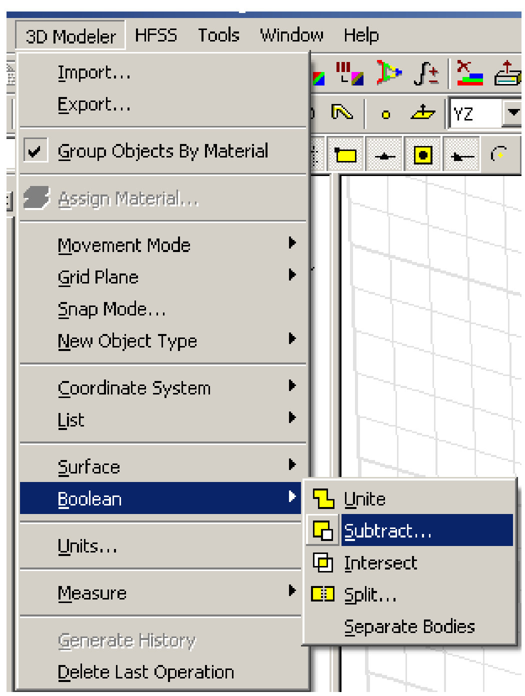


Click OK when complete.

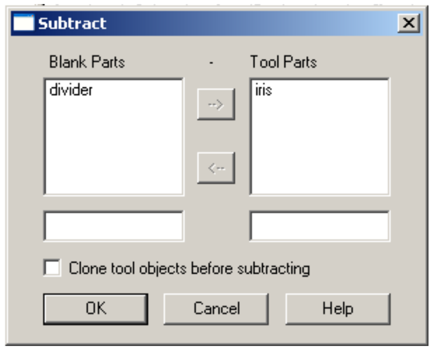
In order to properly create the “iris”, the “iris” must be subtracted from the plate. First, select both objects. Use the Object selection button on the toolbar as shown below:



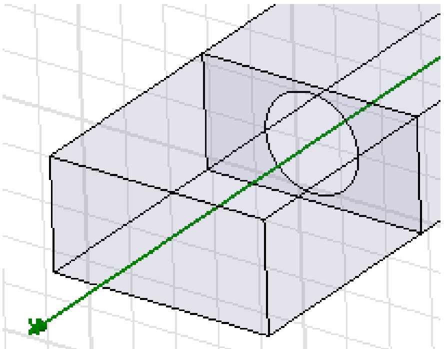
Select both the “divider” and the “iris” and click OK. From the toolbar select 3D Modeler -> Boolean -> Subtract.



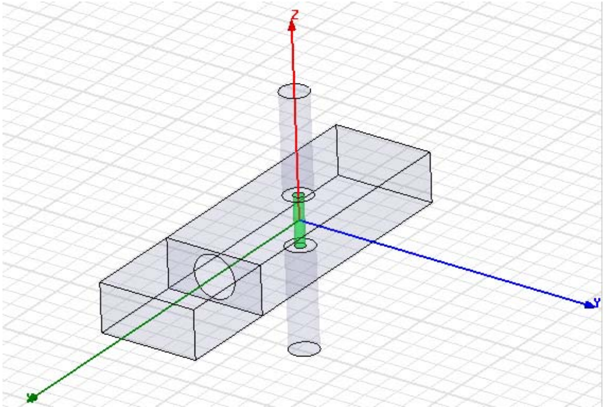
Subtract the “iris” area from the “divider area” as follows:



The “iris” should now resemble the figure below:

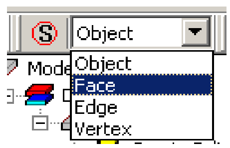


A complete view of the structure is shown below:



Now that the geometry is complete, the user must assign boundary conditions to properly analyze the structure. All external surfaces are defined as Perfect Electric Conductors (PEC) to simulate ideal conductors. Start by using the selection button in the Modeler toolbar.

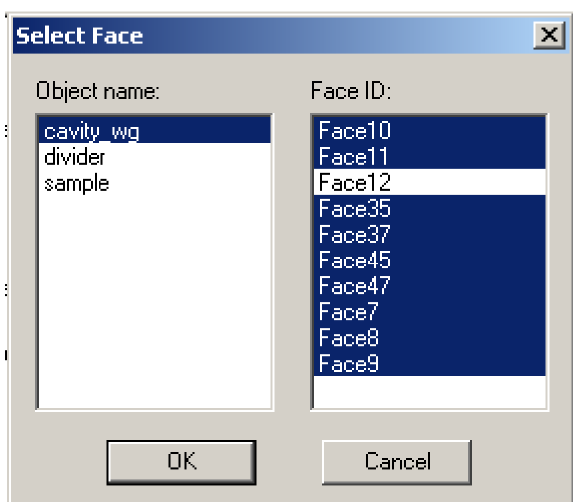
Note: Make sure that Face is selected in the selection edit window.



Select the “cavity_wg” object and highlight the following faces:

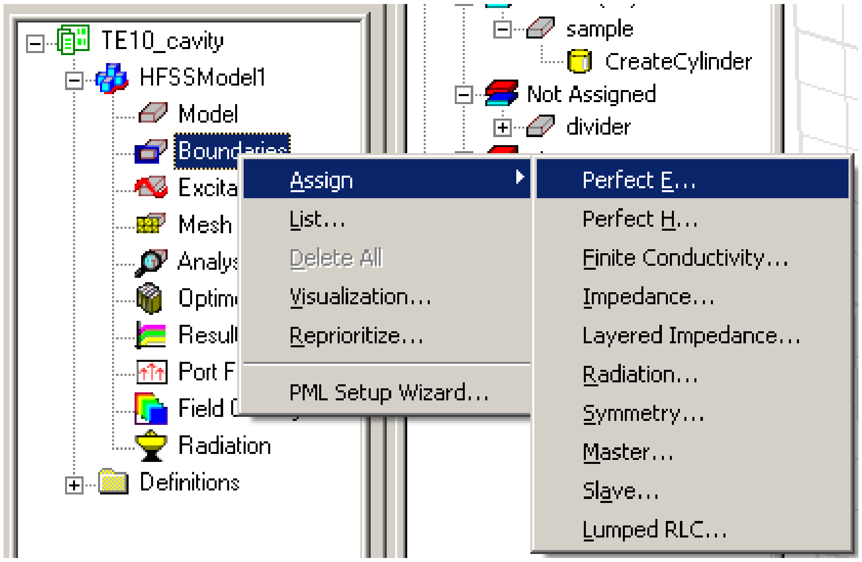
Depending on the order of construction, the user may find that his/her object does not contain Face12. In this case Face12 is the located furthest from the origin along the x-axis and is in the yz plane. Throughout this tutorial, Face12 will refer to this face.

➜ **Warning:**



Notice that “Face12” has not been highlighted. The excitation source will be defined in the next section. This face is used for the source.

From the project tree, Right-Click Boundaries->Assign->Perfect E:



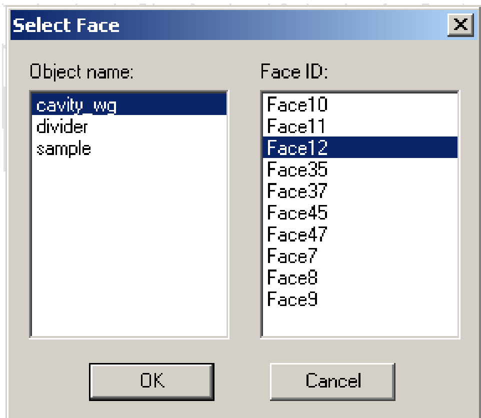
Name the boundary "PEC1". Follow the same procedure and assign the "divider" as "PEC2". All surfaces of the resonating geometry are now defined.

➜ **Note:**

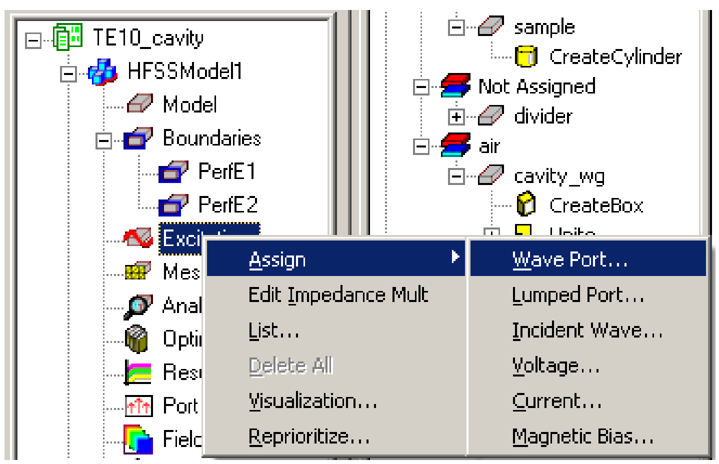
Check the objects and verify the material as "air".

In this example, a wave port will be implemented. The excitation is assigned as a TE_{10} mode implying that the electric field vector is positioned vertically and normal to the axis of the waveguide.

Begin by selecting a face for wave port assignment. Make the following selection:

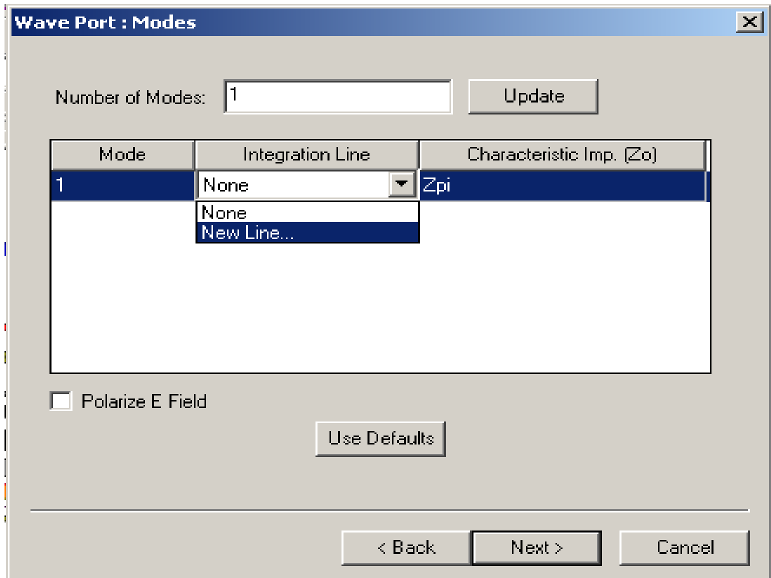


Right-Click Excitation->Assign->WavePort.



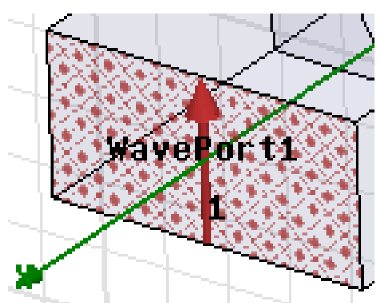
Leave the default name as “WavePort 1”.

Click the modes tab and select New Line... from the drop down menu.

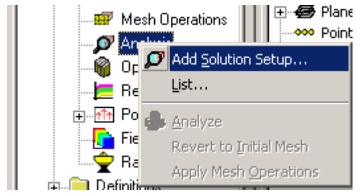


Position the mouse over face12 and let the pointer snap to the bottom center of the plane. Left-Click to define the vector origin. Position the mouse over the top center of face12 and Left-Click to define the endpoint of the vector. Click Next and Finish to complete.

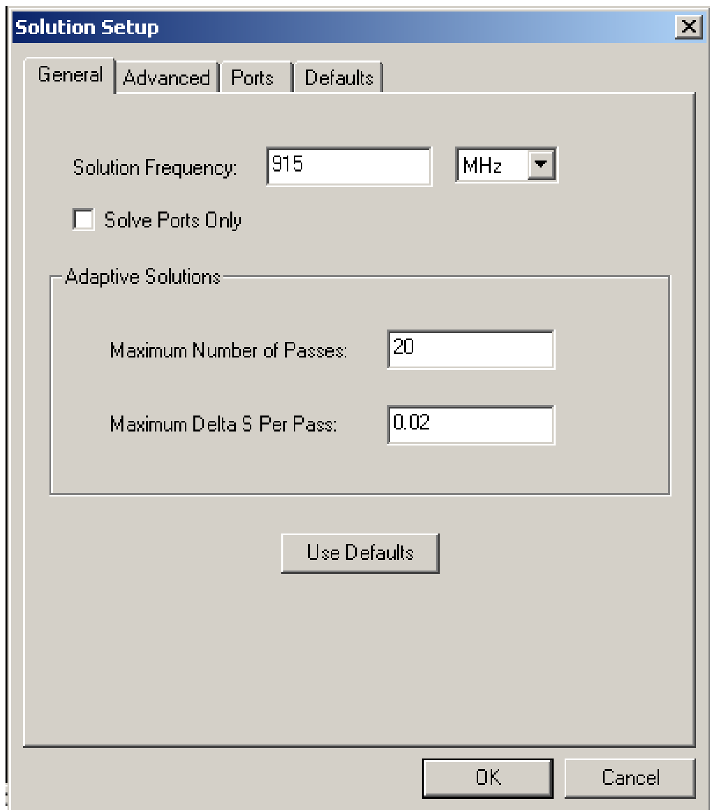
The wave port should resemble the following:



HFSS requires a solution setup in order to analyze the structure. From the project tree, Right-Click Analysis->Add Solution Setup.



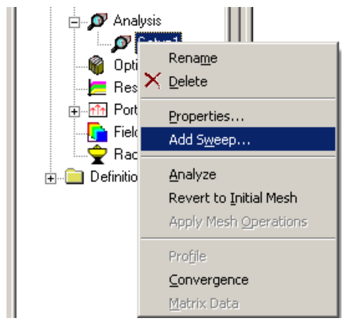
The cavity is expected to resonate at a frequency of 915MHz. Enter this as the solution frequency. Set Maximum Number of Passes to 20 and Delta S to .02.



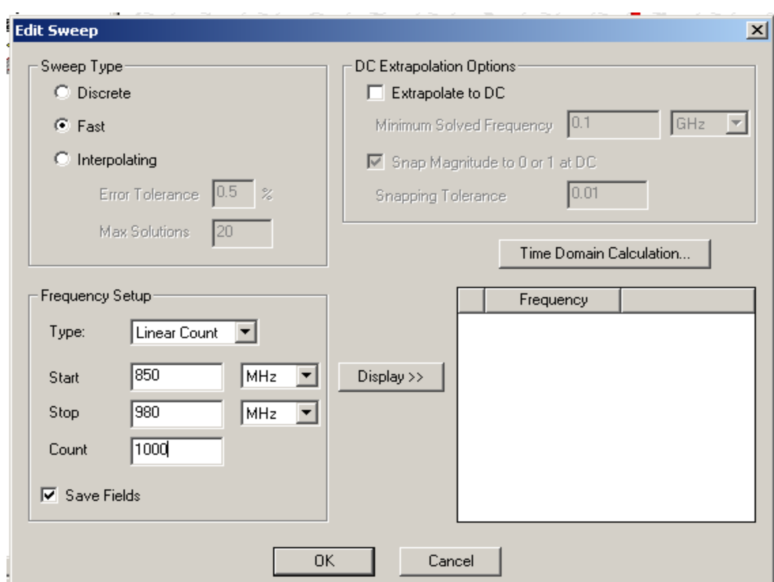
If simulation time is excessive, the user can lessen the computational load by allowing Maximum Delta S >= .05.

→ **Note:**

In order to view the spectrum of S_{11} parameter, HFSS must sweep through a range of frequencies. According to the design, the point of resonance should occur at 915MHz. This will be the center point of the sweep and a tolerance of $^{+}/_{-}$ 65MHz will determine the sweep range. If desired, the user can reanalyze this sweep with less tolerance once the resonant point has been identified. Right-Click Setup1 -> Add Sweep from the window.

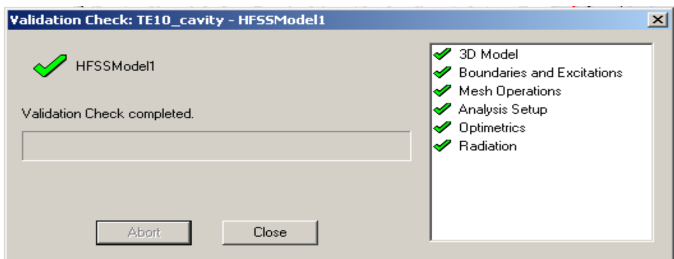


Enter the following parameters in the Edit Sweep window:



Click OK when finished.

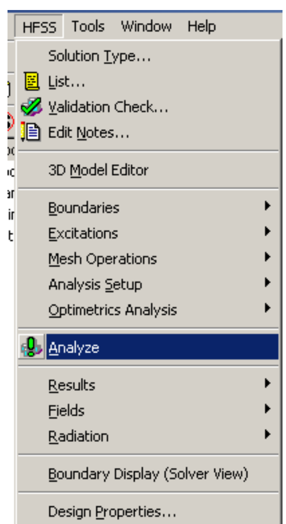
Before starting the Analysis, it is a good idea to run the Validation Checker to verify that the structure and all options have been properly applied. From the toolbar select HFSS->Validation Check.



If errors are present, please correct them before continuing. The results should appear as those in the above figure.

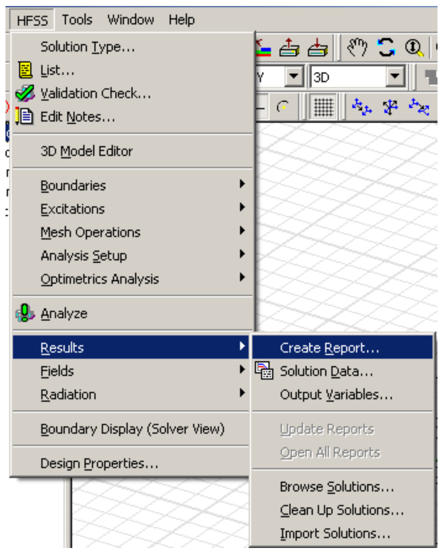
➔ **Note:**

From the toolbar select HFSS -> Analyze.

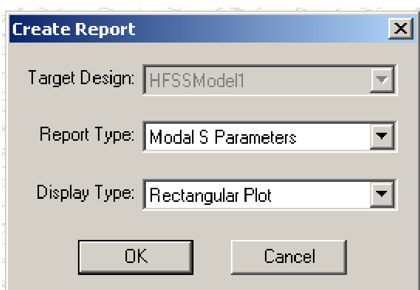


Depending on the machine, the analysis can take from 5-35 minutes.

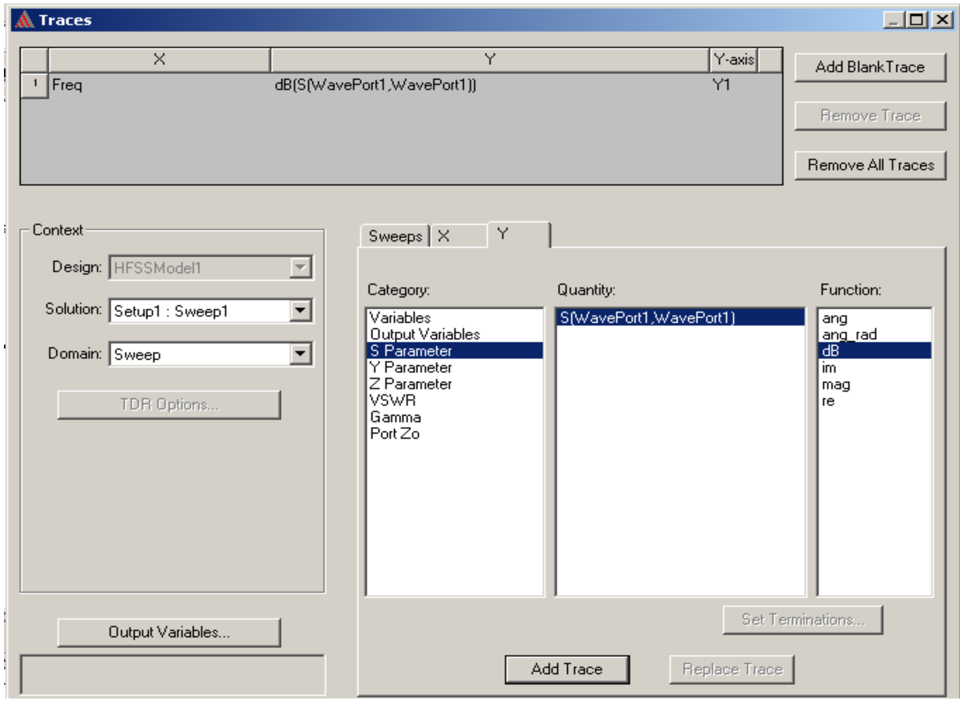
After Analysis, a report must be created to view the desired results from the simulation. From the toolbar select HFSS->Results->Create Report.



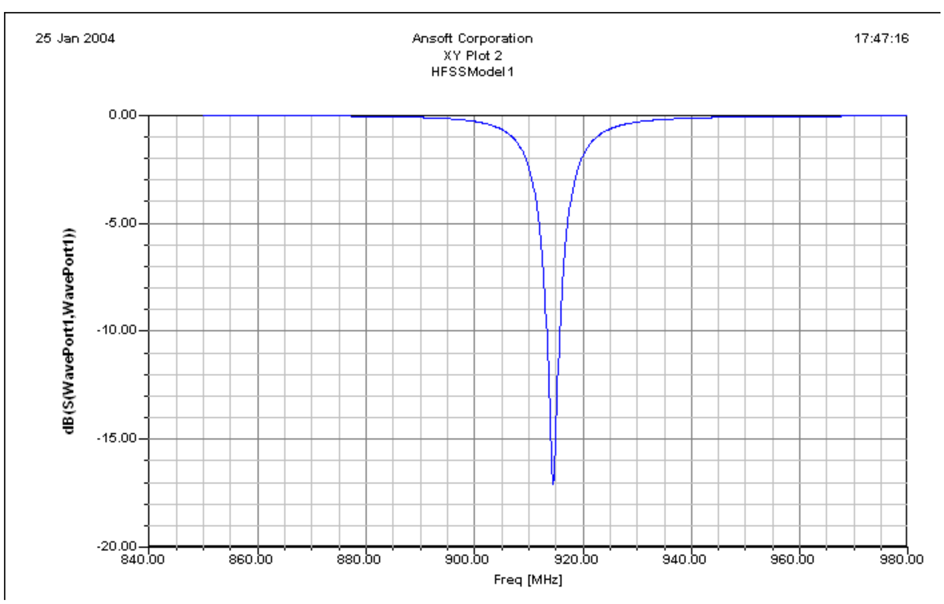
The S-parameter plot will display the reflection coefficient vs. frequency. The resonant frequency can be visually determined from the plot. Enter the following:



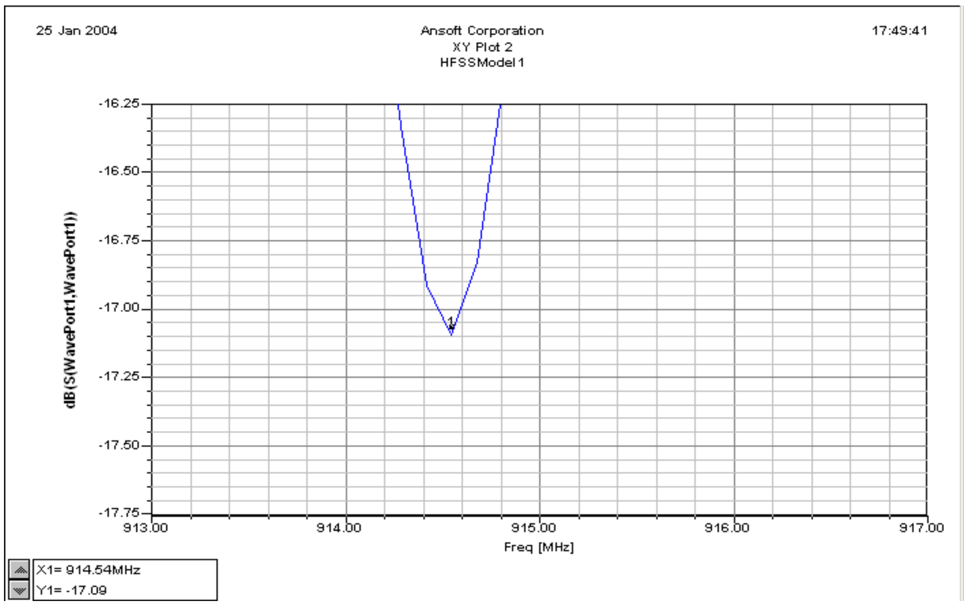
Many options are available from this window. For now, Highlight the variables shown below and click Add Trace. Leave all remaining options as default.



Click Done to complete. The following plot is displayed:



Inspection of the previous figure suggests that the resonant frequency is approximately 915MHz. Use the zoom feature by Right-Clicking the plot area and select Zoom In. Place a window around the peak. Right-Click the plot area and select Data Marker . Select the lowest point to determine the resonant frequency. The plot is shown below:



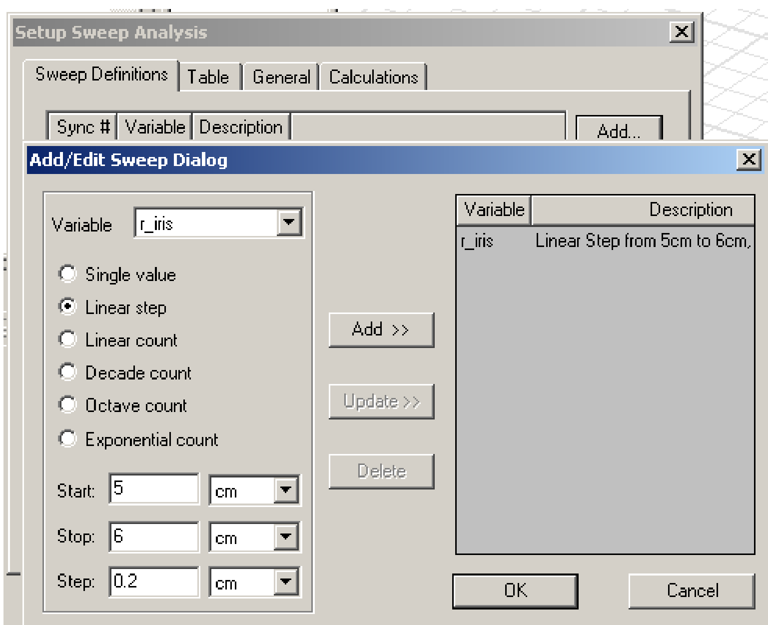
The resolution in this analysis is a limiting factor. Resolution is increased by narrowing the sweep range and keep the count size constant. The user can also explore other types of sweeps (see Ansoft help).

Note:

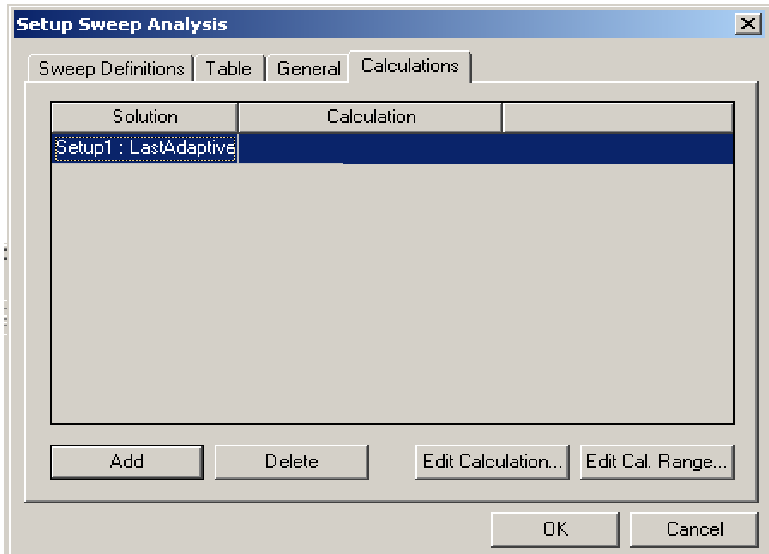
In HFSS the optimetric tool is used to optimize variables according to user defined input. Variables are optimized and this could save the designer hours of trial and error using empirical and/or experimental approach. In this tutorial, all variables were given to the user. Let us assume, as a designer, that the only adjustable variable in the design is the “iris”. All other values are fixed. We will optimize the radius for maximum power transfer, at a frequency of 915MHz, into the cavity. According to experimental data, the optimum radius should lie between 5-6cm. Parametric analysis will allow calculation of fields for different values of “r_iris”.

From the project tree select Optimetrics -> Add -> Parametric.

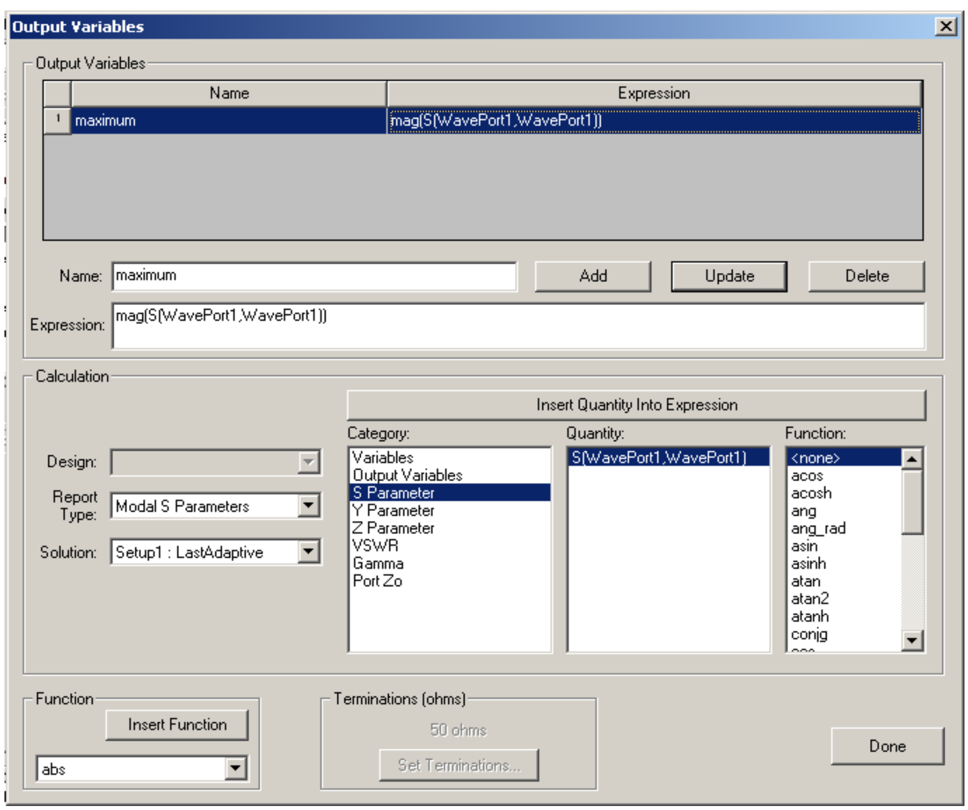
Select “r_iris” as the variable. The radius will begin at 5cm and end at 6cm with a step size of .2cm. Click Add and OK when finished.



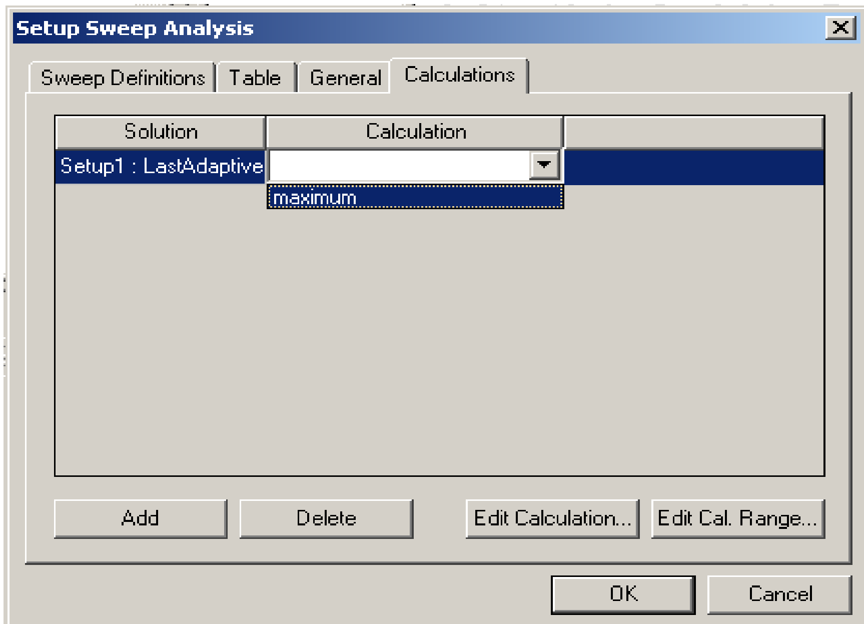
From the Calculations tab, select Edit Calculation.



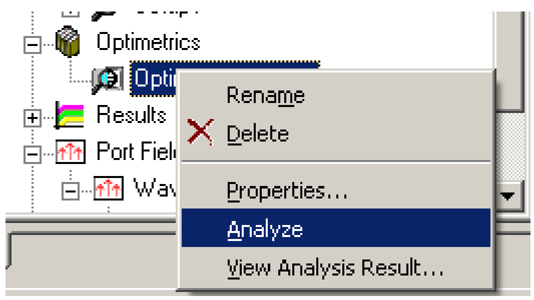
We will examine the S_{11} parameters so enter the following power expression:



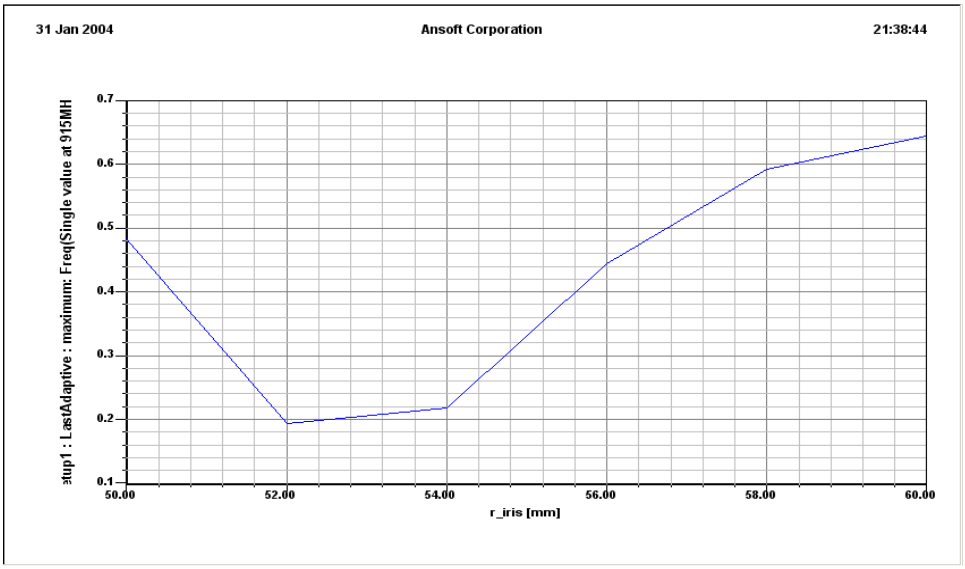
Name the expression "maximum". Click Add and Done when complete. Under the calculation edit window, select "maximum" from the drop down menu as shown below:



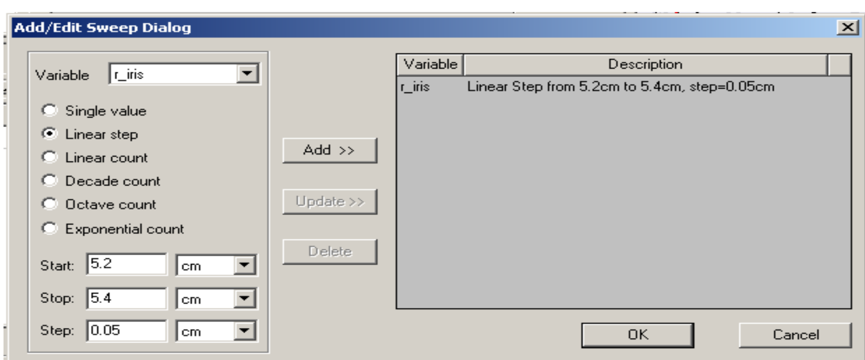
Click OK when complete. From the project tree, right-click Optimetrics -> Analyze. Depending on the machine, analysis can take from 5 minutes to 1 hour.



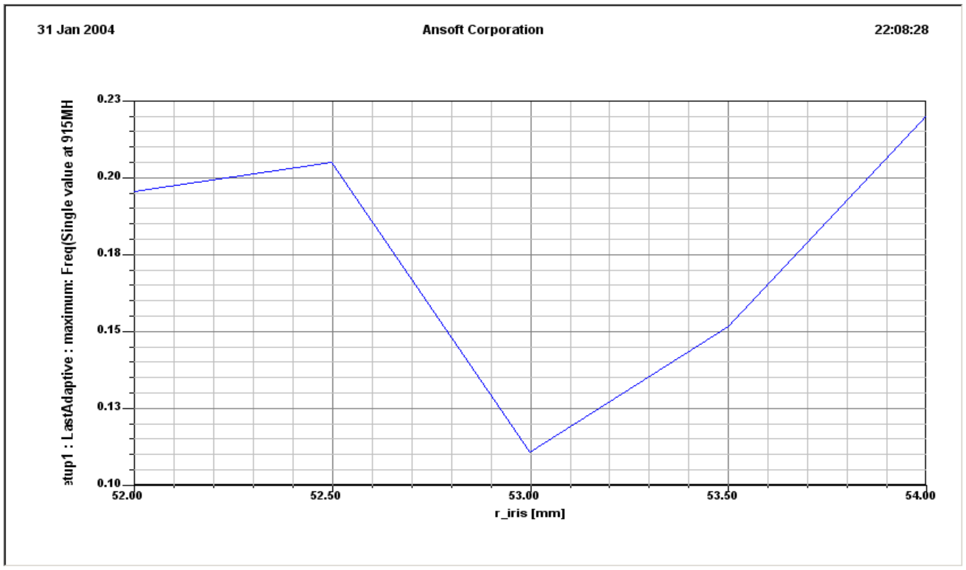
With analysis complete, Right-Click Optimetrics -> View Analysis Result and select plot in the window. You should see the following results.



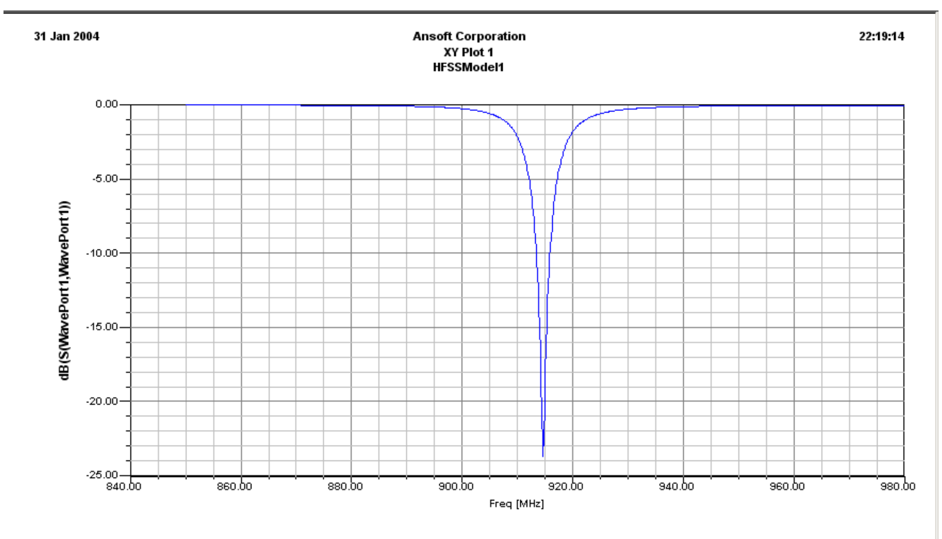
The above figure displays S_{11} vs. iris radius. We find that minimum reflection occurs between 5.2cm and 5.4cm. With this information, we can now re-analyze the iris variable defined from 5.2cm to 5.4cm. Set the step size to .05cm. Follow all previous step and re-analyze with the new settings.



When complete, view the S_{11} parameter from the plot. The results are shown below:

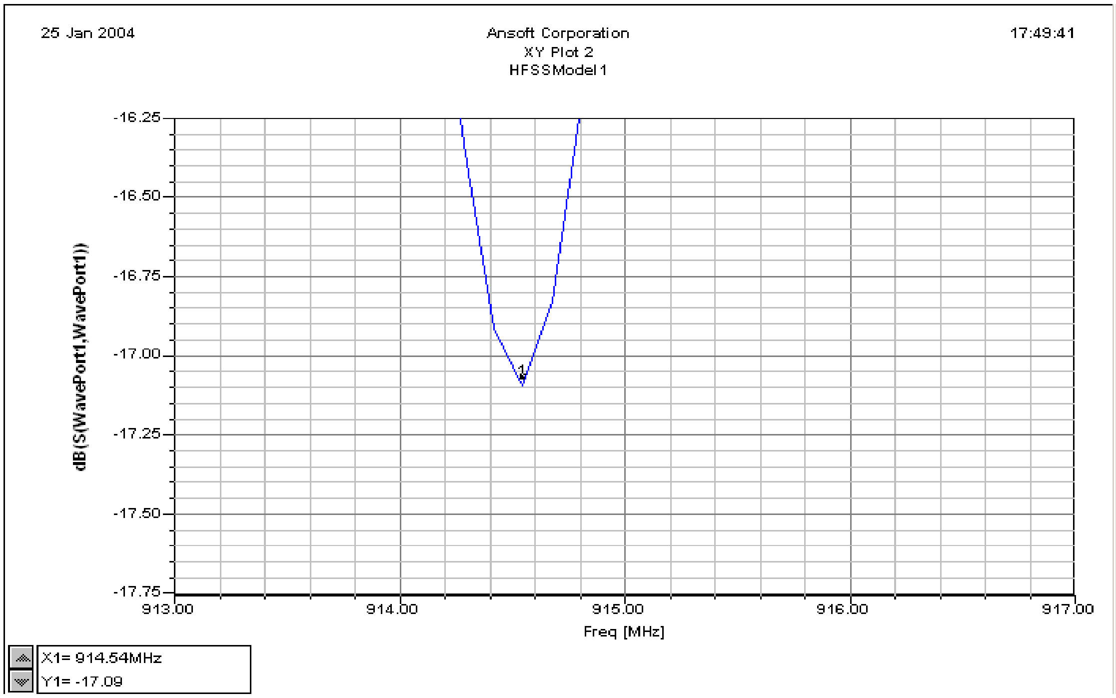


According to the graph, minimum reflection occurs at approximately 5.3cm. This should produce optimal power transfer into the cavity. With this information, redefine the "r_iris" variable under HFSS -> Design Variables. Follow all previously completed steps in this tutorial and rerun the Analysis with the new variable definition. Create Reports and generate S11 plot. What can you determine from the new plot ?

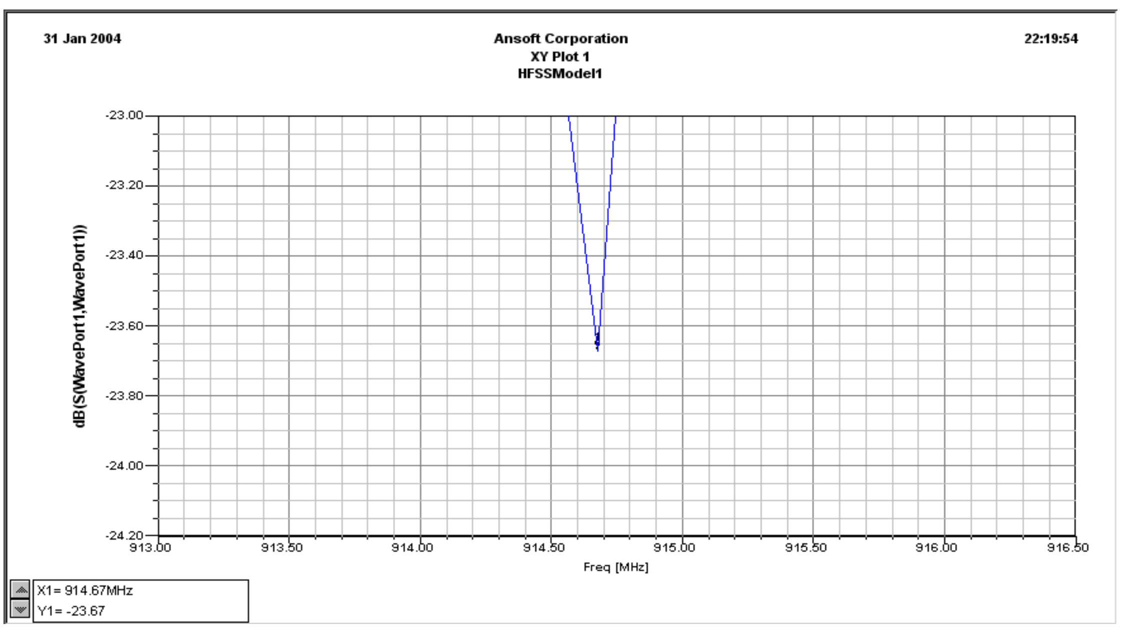


Comparing both S_{11} plots should verify results. See the figures below:

Original : r_iris = 5.3634cm

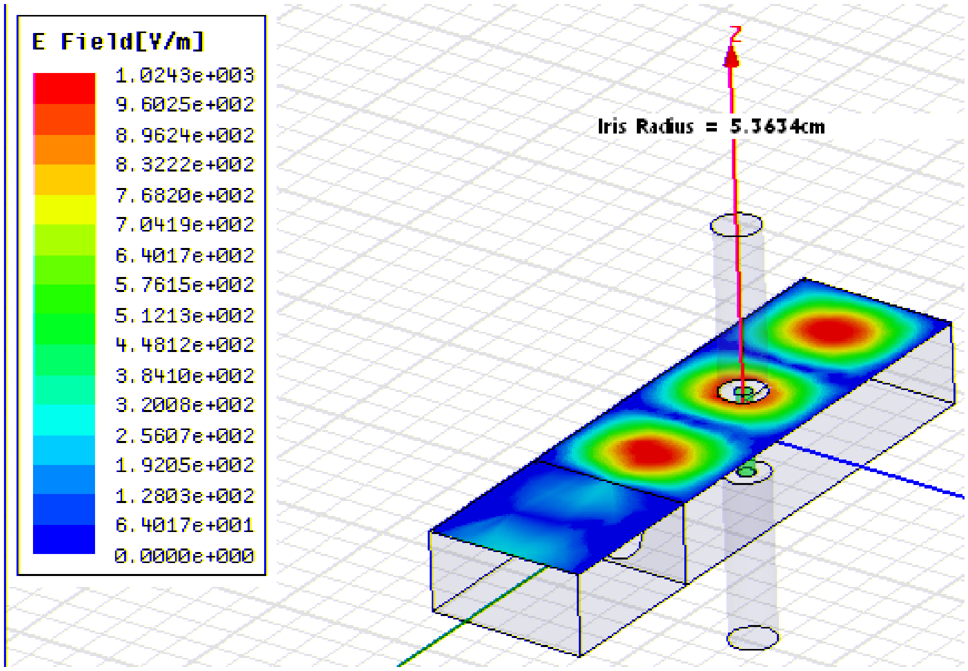


Optimized : r_iris = 5.3cm

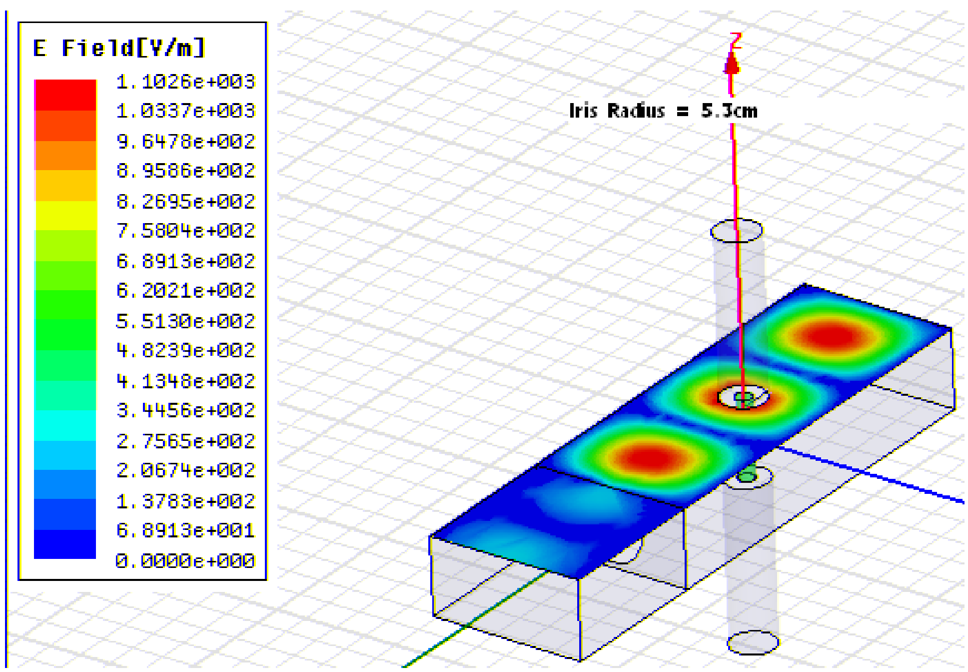


In addition to the S11 plots, the E-field is plotted along the length of the cavity. Note the maximum Electric Field in the plots below:

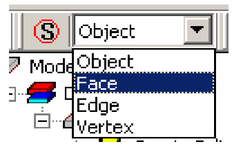
Before Optimization (5.363cm)



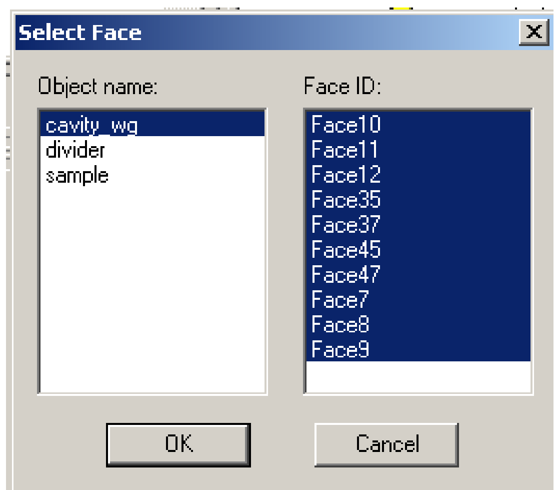
After Optimization (5.3cm)



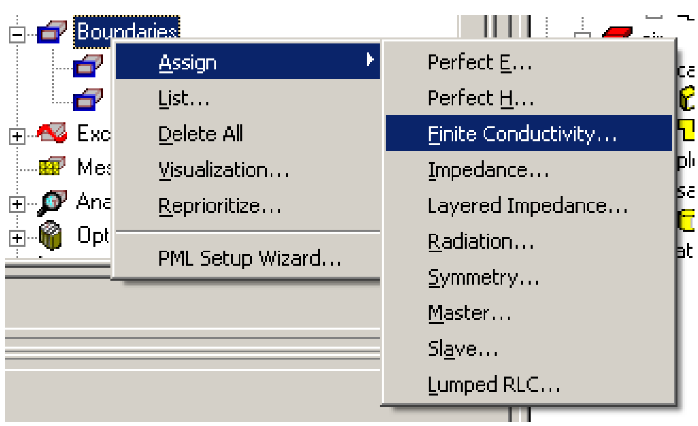
In this section, the PEC material will be changed to Aluminum. Results obtained should be more realistic for obvious reasons. From the toolbar select Face from the select object bar.



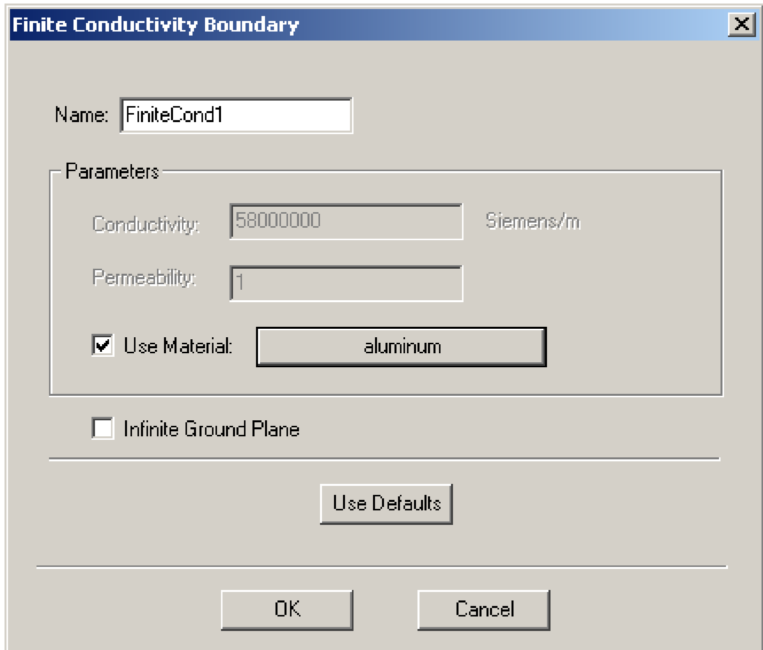
Select all faces belonging to "cavity_wg" as shown below:



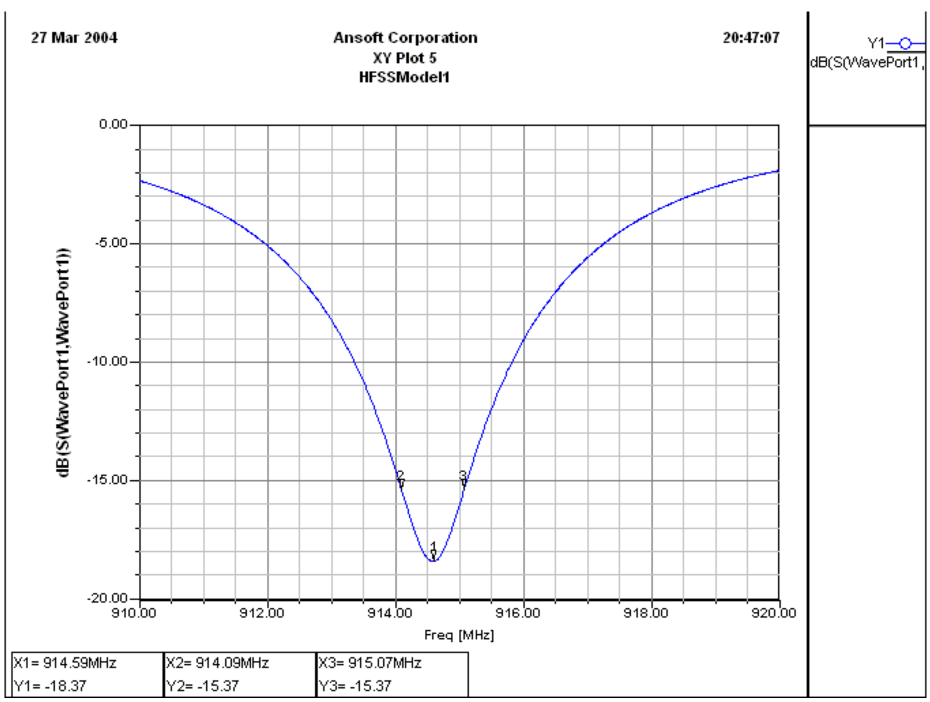
From the Project Manager window, right-click Boundaries -> Assign -> Finite Conductivity.



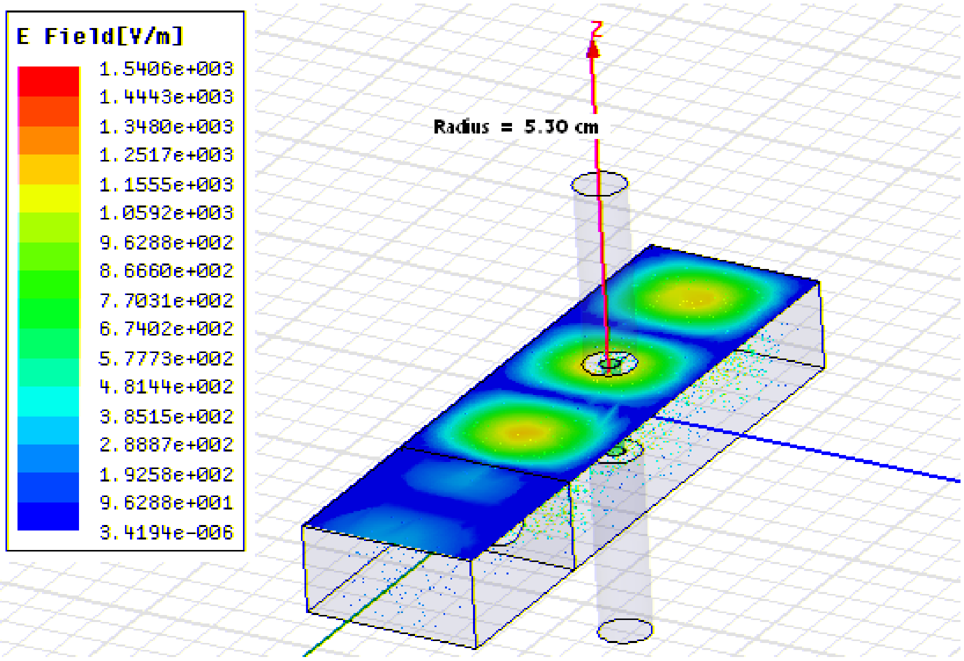
From the *Finite Conductivity Boundary* box, select Aluminum and click OK.



Save your work. Analyze the structure as performed with the PEC structure. When complete, plot the S_{11} parameter. The results are displayed below:

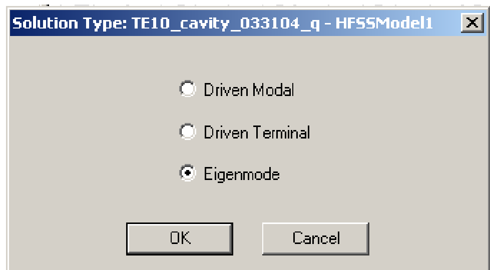


The E-field plot for Aluminum is given below:



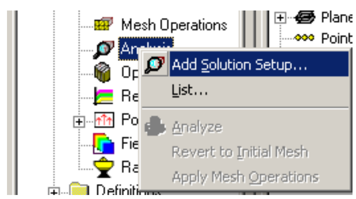
Optimetrics should be used from this point to re-optimize the iris for maximum power transfer. This will be left as an exercise for the user.

In order to determine the Quality Factor of the circuit, the solution type must be changed to Eigenmode. From the HFSS toolbar select:

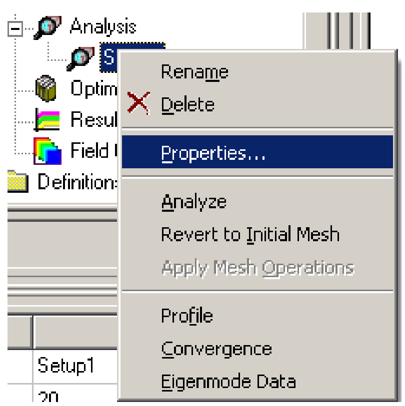


Click OK when complete.

From the Project Manager window, Create a solution setup as follows:

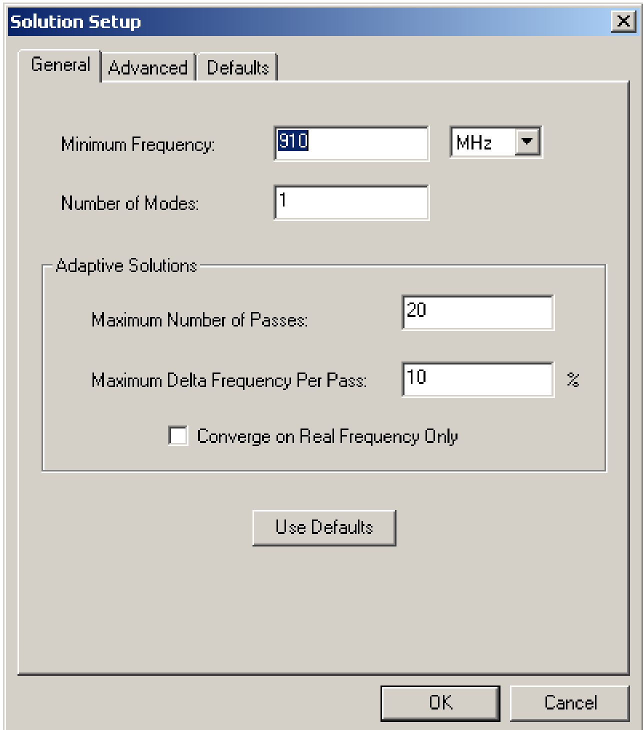


Right-click Analysis -> Add Solution Setup.

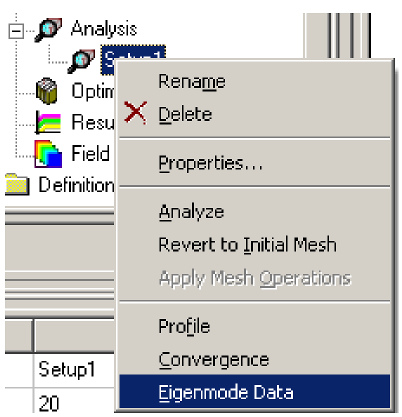


Right-click Setup1 -> Properties.

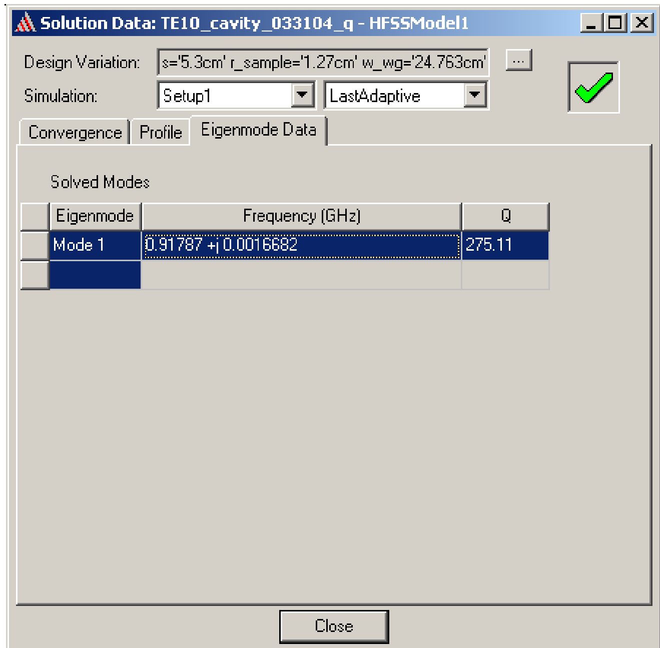
Set Minimum Frequency = 910 MHz. Click OK when complete.



The eigenmode solution will find the cavity resonance and the Q of the cavity. Run the Validation Check as in the previous example to verify correct setup. Run the Analysis. When complete, right-click Setup1-> EigenmodeData.



The resonant frequency and quality factor are given as shown below:



The Eigenmode solution found the resonant frequency at 917MHz. This calculation could be made more accurate by decreasing the Maximum Delta Frequency Per Pass within the analysis setup.

Conclusion

This tutorial should give the user an introduction to HFSS and a few of the available features. The user should now be able to begin designing and optimizing structures on his/her own.