Numerical Modeling of an Octagonal Waveguide Applicator for High Power Microwave Heating Applications Using Ansoft's HFSS

The objective of this tutorial is to analyze an octagonal-shaped waveguide applicator with two cylindrical cutoff tubes located at the center of the top and bottom faces of the applicator. The applicator is designed to operate in the L-band. Our objective is to identify the resonant frequency at which the applicator will operate and to evaluate its performance when loaded with various lossy dielectric loads.

The applicator consists of eight rectangles, three rectangles at the top and bottom, and one on each side, and two cylinders, as depicted in Figure 1.

.

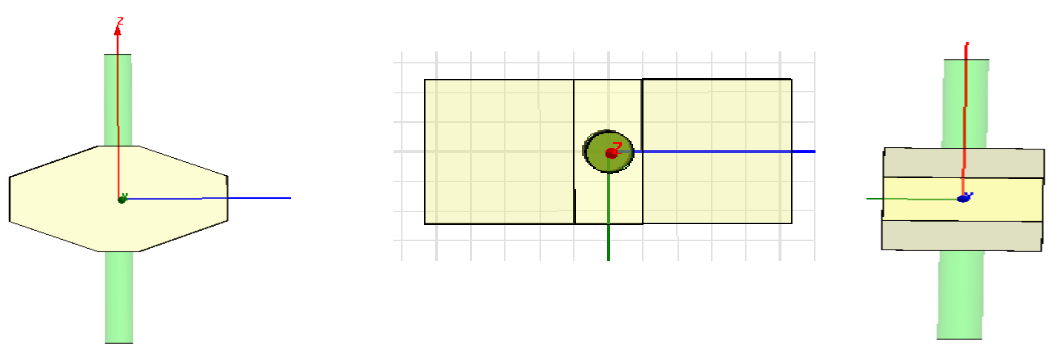

Figure 1: Front, top and side view, respectively, of the octagonal waveguide applicator

The dimensions are as follows:

Cylinders: Diameter = 2.75 inches and height = 10 inches.

Side Rectangles: Length = 10.125 inches and width = 4.875 inches.

Top Rectangles: Length = 4 inches and width = 9.75 inches.

Inclined Rectangles: Length = 9.28 inches, width = 9.75 inches, inclined at an angle of 21° with respect to the horizontal.

Open HFSS and save a New Project

How to start HFSS?

HFSS should be installed on the computer or station you are working on. There should be an HFSS icon on the desktop you can double click on it to launch HFSS, or you can go to "**Start**" button on the lower left corner of your screen, click on it, then go to "**Programs**" button and a list of programs will pop up. Go to "**Ansoft<< HFSS 9<< HFSS 9**".

How to Open a New Project?

As you start HFSS v 9, a project is listed in the project tree in the **Project Manager** window and is named project1 by default. Project definitions, such as material assignment, boundary conditions, and port excitations are stored under the project name.

Save the Project

On the **File** menu, click **Save As.** Use the file browser to locate the folder in which you want to save the project, such as C:\Ansoft\HFSS9\Projects, and then double-click the folder's name. Type **WR975** in the **File Name** text box and click **Save**. Now, the project is saved in the folder you selected by the file name with an extension of hfss, **WR975.hfss.**

Insert an HFSS Design

On the **Project** menu, click **Insert HFSS Design**. The new design is listed in the project tree. It is named Model by default. The **3D Modeler** window appears to the right of the Project Manager.

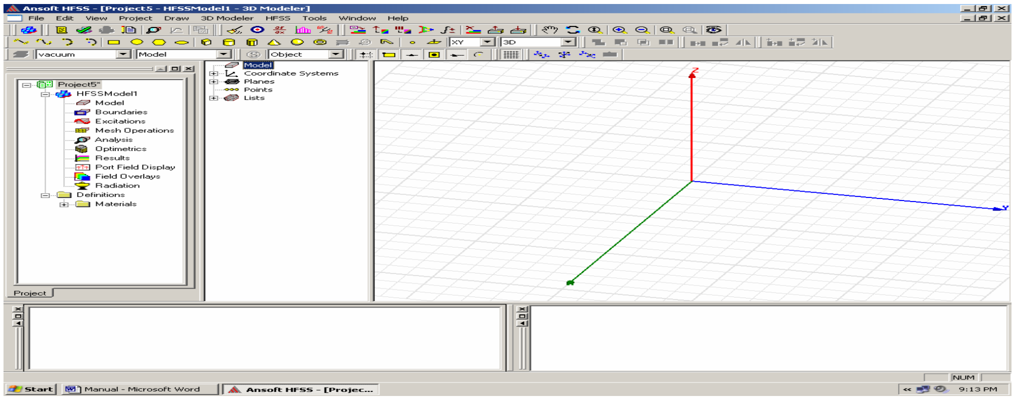

Figure 2: HFSS window

You can rename the model by right-clicking on the Model and then click **Rename**. Type the name then press **Enter**.

Select a Solution type

Now you need to specify the design's solution type. As you set up the design for analysis, available settings will depend upon the solution type. For this design, you will choose **Driven Modal** as the solution type, which is appropriate when calculating mode-based *S*-parameters of a passive device that is being driven by a source.

On the **HFSS** menu, click **Solution Type**. In the **Solution type** dialog box, select **Driven Modal**, and then click **OK.**

Set the Drawing Units

You will now set the units of measurement for drawing the geometric model.

On the **3D Modeler** menu, click **Units.** In the **Set Model Units** dialog box, click in the **Select Units** pull-down list, and then click **OK.**

Creating the 3-D model

The first step in simulating any design in HFSS is to draw the geometry of the structure under consideration in 3-D. In the case of the current design, these steps should be followed:

Drawing rectangles:

The user will start by drawing the three rectangles that will constitute the upper side of the cavity.

- The first rectangle will have the following starting point coordinates (-2,-10.125/2, 11.5/2) with 4 as its *X*-size and 10.125 as its *Y*-size.
- The second rectangle will have (0, 0, 0) as starting point coordinates, with 9.28 as its *X*-size and 10.125 as its *Y*-size.
- The third rectangle will have (0, 0, 0) as starting point coordinates, with -9.28 as its *X*-size and 10.125 as its *Y*-size.

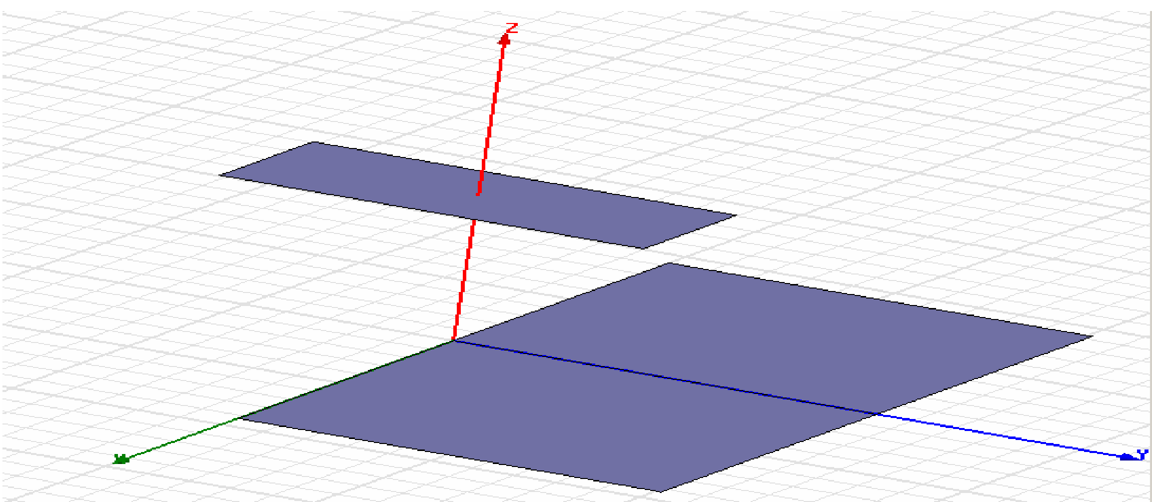

Figure 3: The three rectangles.

To draw a rectangle, the user should first left click on the **draw rectangle** icon  in the toolbar, next left click on the drawing area and drag, and then left click again. This will cause a new window to appear where the user will enter the coordinates of the starting point of the rectangle, and its dimensions.

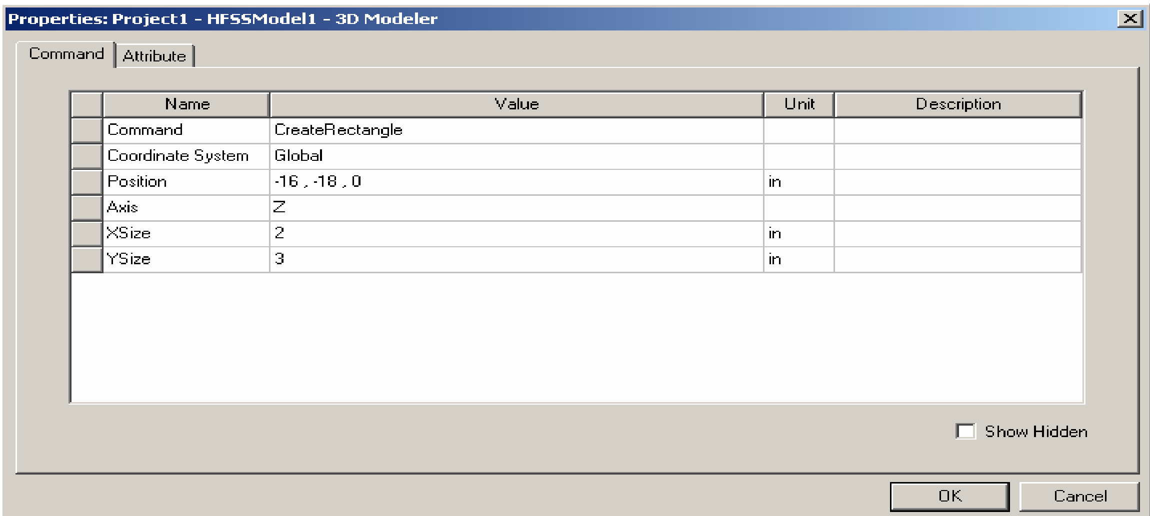

Figure 4: Rectangle coordinates window

Rotate:

Now rotate the second and third rectangles by 21 and -21 degrees respectively, with respect to the *Y*-axis. To rotate, select the object then right click on it and press on **Edit>Arrange>Rotate**.

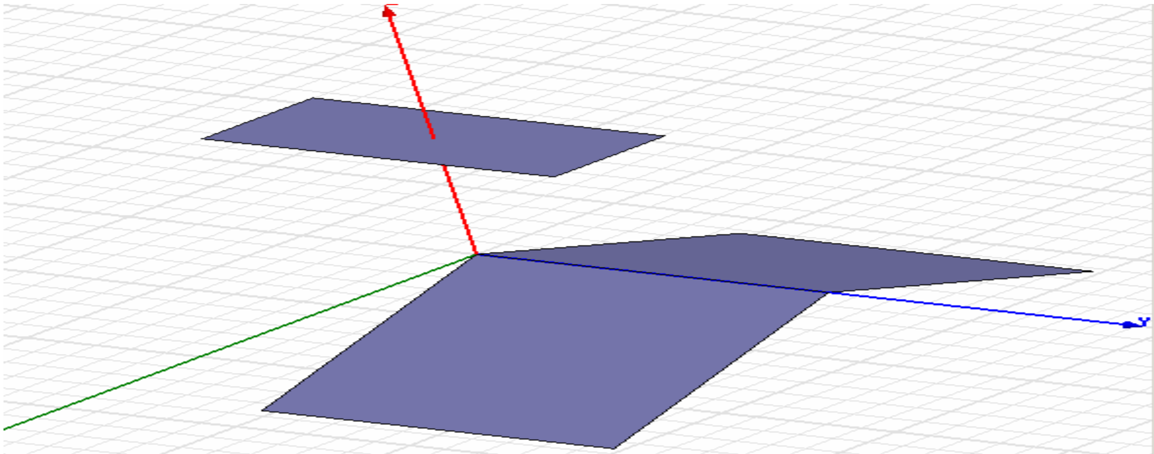

Figure 5: After rotation

Move:

Now move the two rotated rectangles to there corresponding positions (adjacent to the first rectangle). To move an object, first select it, next right click on it and press on **Edit>Arrange>Move**.

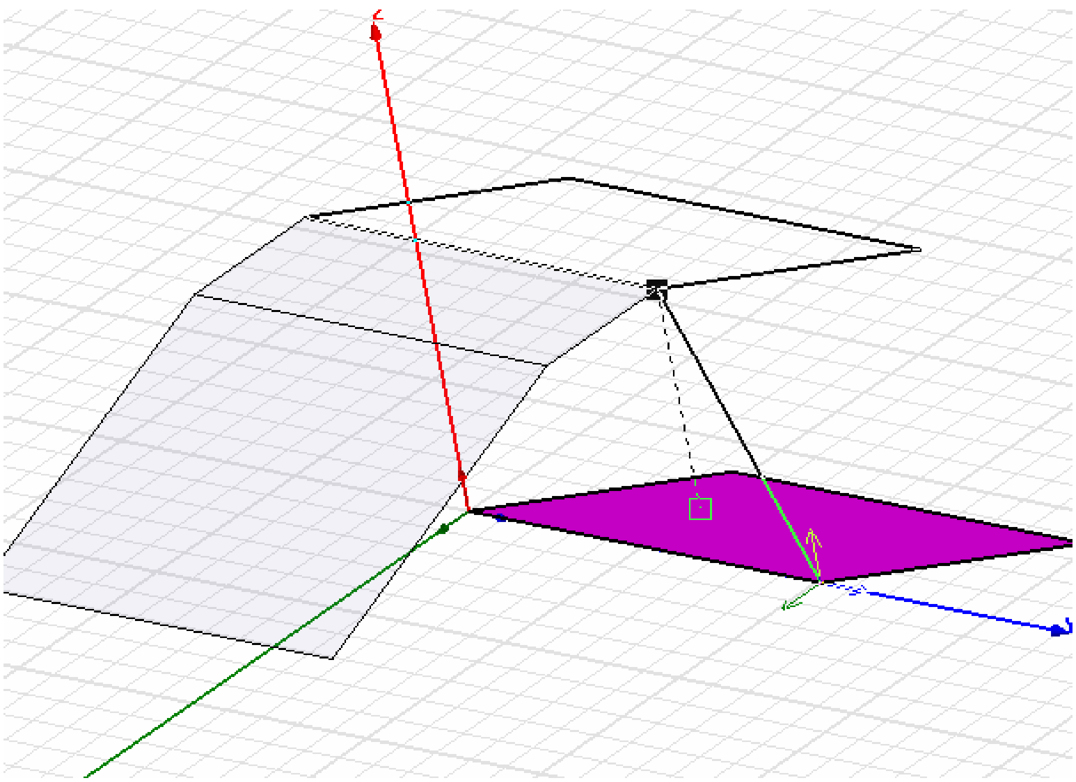

Figure 6: Applying the move command

Duplicate:

With the upper wall now created, we will use the symmetry properties of the design to draw the lower wall of the cavity. Select the upper wall, right click and press on **Edit > Duplicate > Along axis** (To choose multiple objects at the same time, left click on the objects while pressing the Ctrl key). A new window will appear. Select the *X*-axis and change the angle to 180 degrees. This will cause the lower part of the cavity to be created.

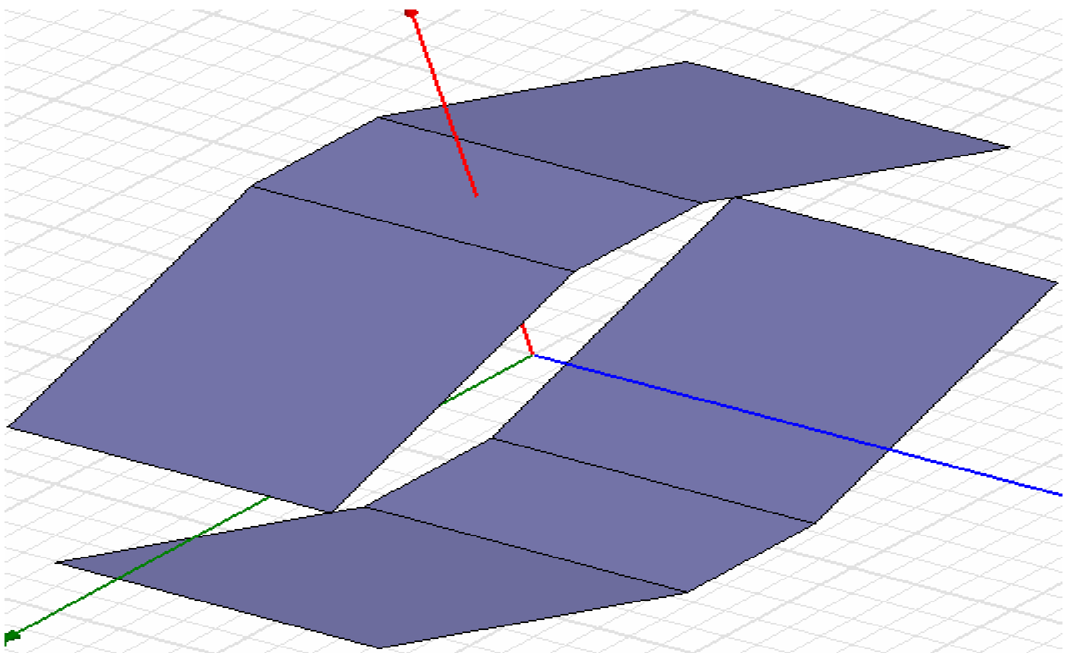

Figure 7: After duplication

Connect:

To create the sides of the cavity, we will simply connect the upper and lower sides that were already created. Select each one of the three rectangles and its duplicate. Right click and press on **Edit > Surface > Connect**. Repeat the preceding step for the other two rectangles.

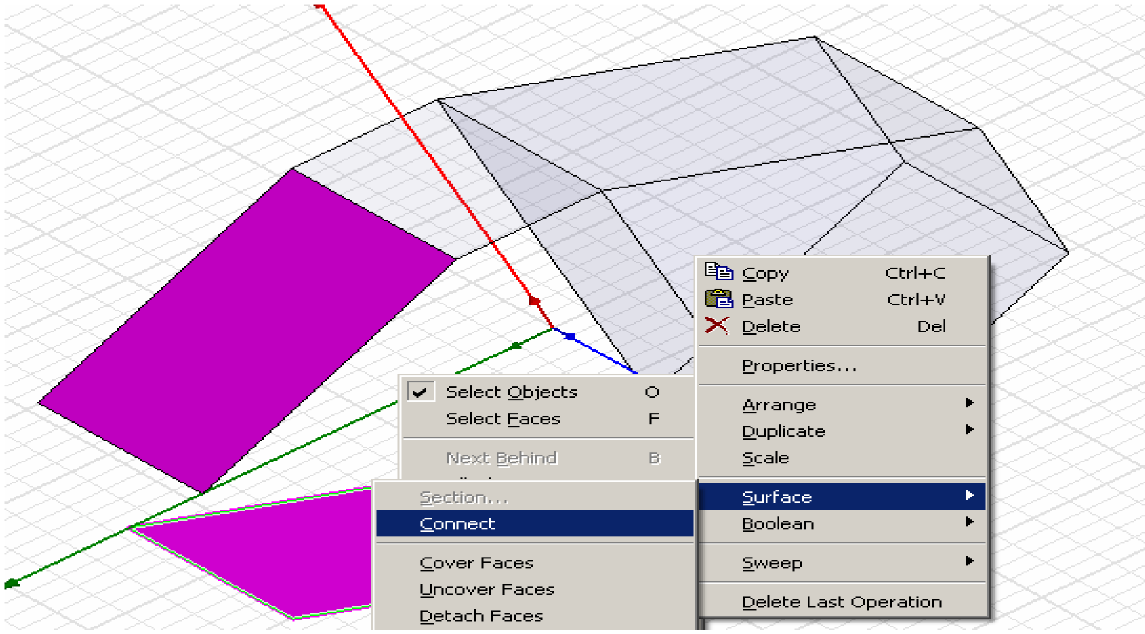

Figure 8: Applying the connect command

Unite:

This step is used to eliminate the common sides formed when we used the connect function. By uniting the three objects that were created we will get one continuous object. This is done by selecting all three objects, right clicking and pressing on **Edit>Boolean>Unite**.

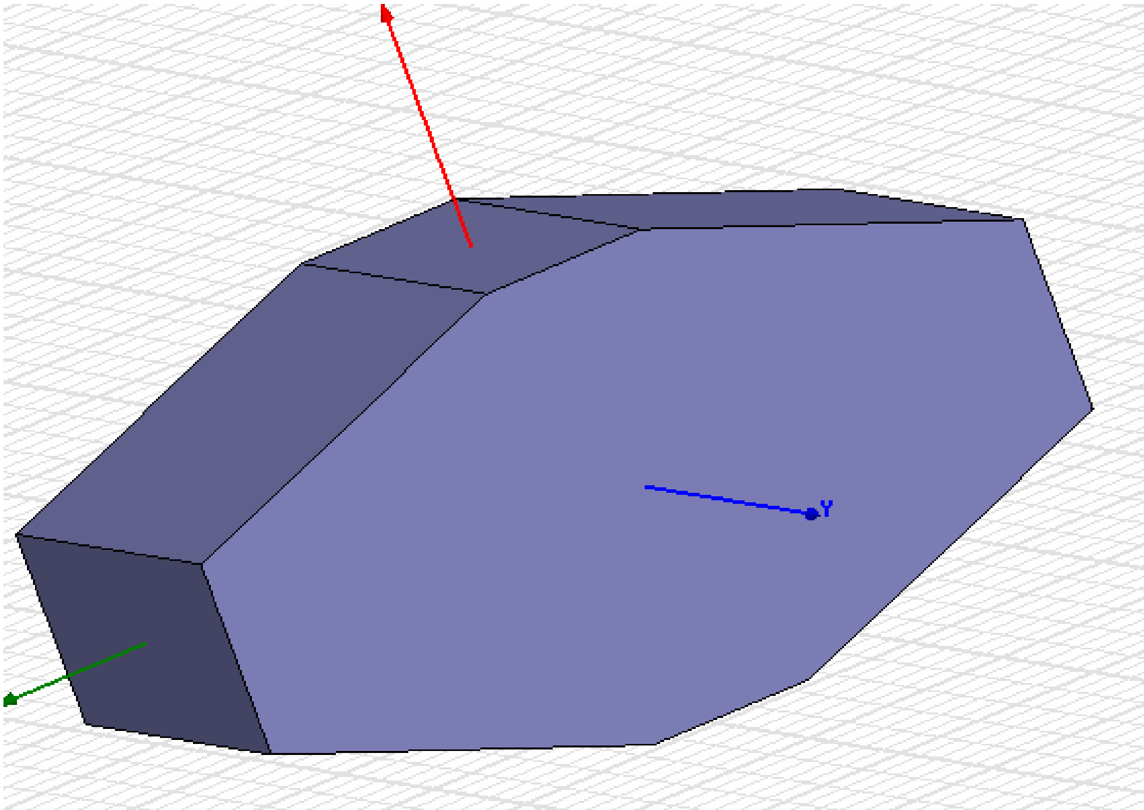

Figure 9: Object after being united

Drawing cylinders:

The last step is to draw the two cylinders that will make up the openings through which the object to be heated will be introduced. This is done by clicking on the draw a cylinder icon  and entering the following dimensions:

- First cylinder has (0, 0, 11.5/2) as its center coordinates, 10 as its height, and 2.75/2 as its radius.
- Second cylinder has (0, 0, -11.5/2) as its center coordinates, -10 as its height, and 2.75/2 as its radius.

Finally, unite the two cylinders with the original object in order to eliminate the intersecting areas.

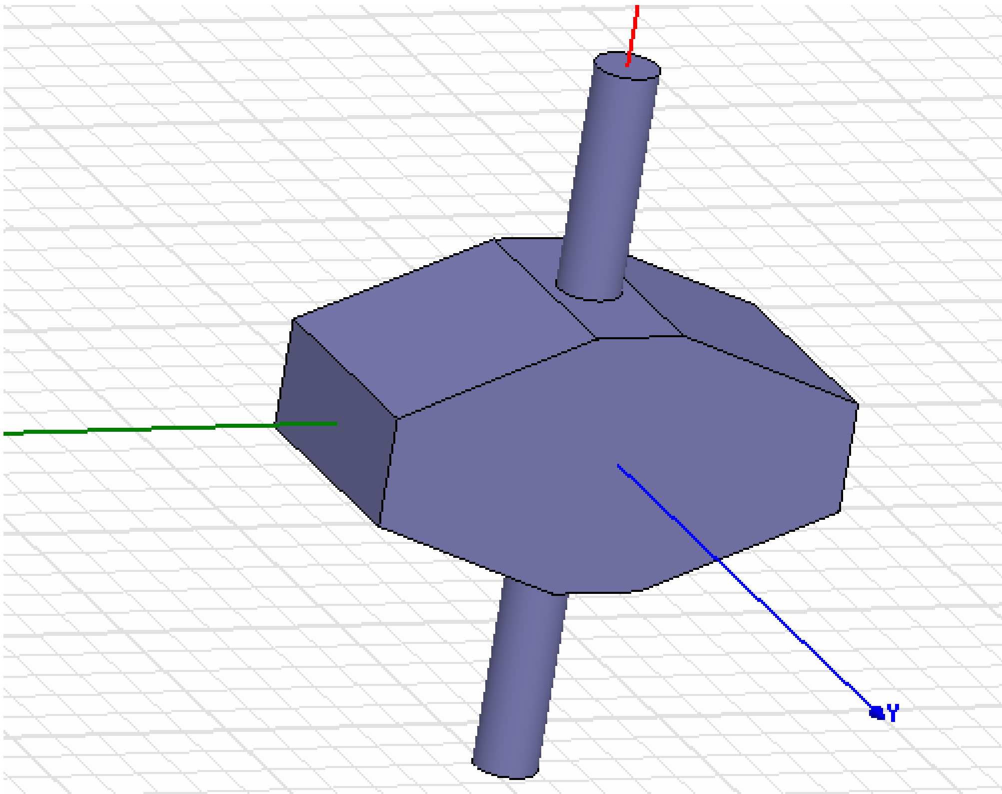

Figure 10: The final 3-D model

Boundary conditions and excitation assignment

Having built the model, the user will now have to assign the boundary conditions. To assign boundary conditions, start by choosing the faces of the object created to which the same boundary condition will be applied (in our case these are all but three of the object's faces). To select multiple faces of an object at the same time select **Face** in the selection edit window then press on the  icon. A window will appear with the list of faces belonging to the object that was created. While pressing down the Ctrl button left click on the names of the faces you want to select. When selected, the face as well as its name will be highlighted. In our design we want to select all the faces except the two circular ones as well as the front most one. After selecting these faces, right click and press on assign boundary and choose PEC (We are assuming these faces to be perfect electric conductors thus neglecting the finite conductivity due to the cavity walls).

Next is the source assignment. Select the front most side of the cavity (the only non-circular side that was not selected before), right click and choose **Assign excitation>Wave Port**, a window will appear where nothing needs to be changed, just press next.

Solution Setup

The user is now required to add a solution setup. This step involves specifying the frequency and the accuracy at which the user wants the design to be simulated at and is done by choosing **HFSS>Analysis>Add Solution Setup**. This will cause a new window to appear, in which the user will enter 915 MHz as the **solution frequency**, 10 as the **maximum number of passes**, and 0.02 as the **maximum delta S per pass**. Note that the user may vary the number of passes and the value of delta S knowing that these are proportional and inversely proportional to the accuracy of the results and the computational time, respectively.

Since it is preferred to compare the value of S_{11} at the solution frequency with its value at neighboring frequencies, the user will sweep through a range of frequencies, and this is done by pressing on **HFSS>Analysis>Add Sweep**. The user will enter the parameters shown in Figure 11 in the window that appears.

Finally, it is always useful to run the **Validation Checker** before analyzing. This will point out any errors that may have been committed. This is done by pressing on the  icon from the toolbar. If no errors appear then the user can proceed in analyzing his/her design by pressing on the  from the toolbar.

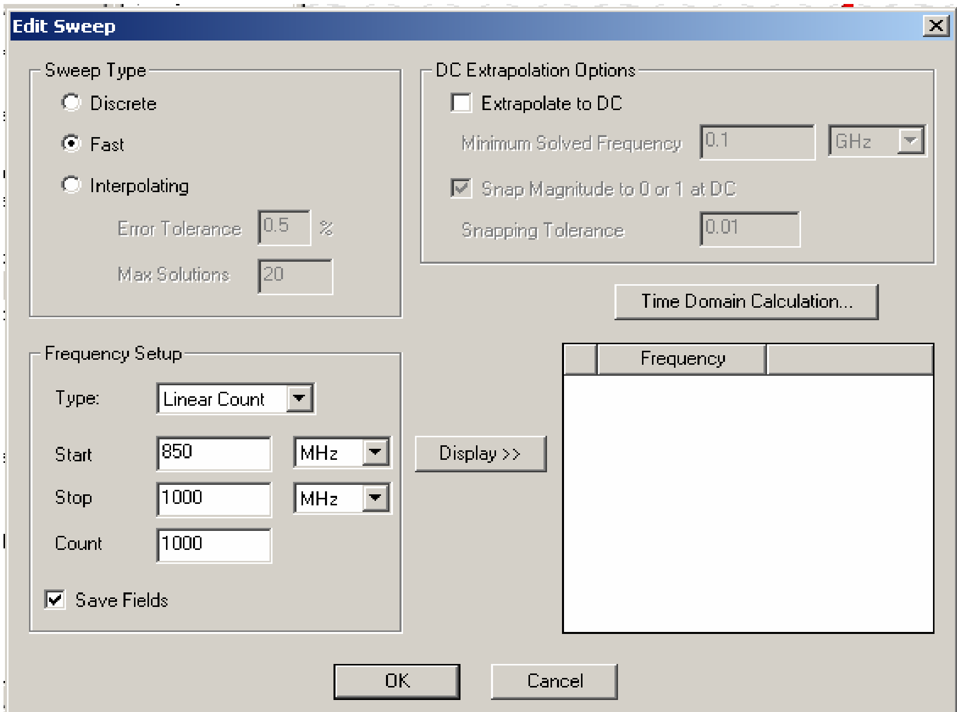

Figure 11: Sweep window

Creating the report

Once the analysis is complete, we need to view the results of the simulation. To see the different values of S_{11}, select **HFSS>Results>Create Report**. Enter the following parameters in the two windows that will follow.

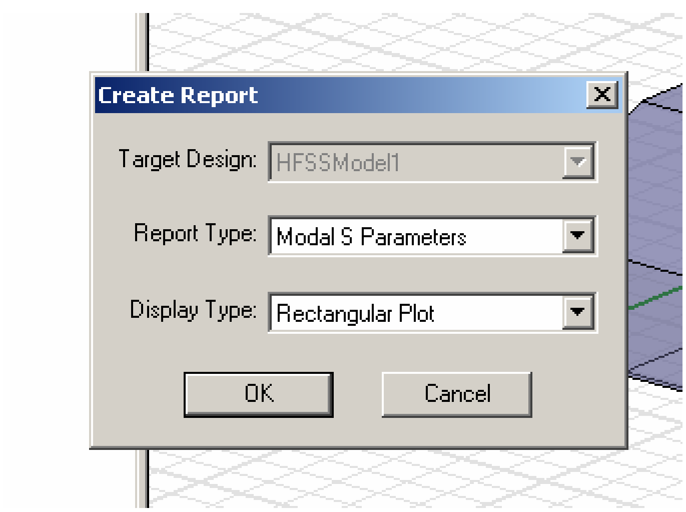

Figure 12: Create report window

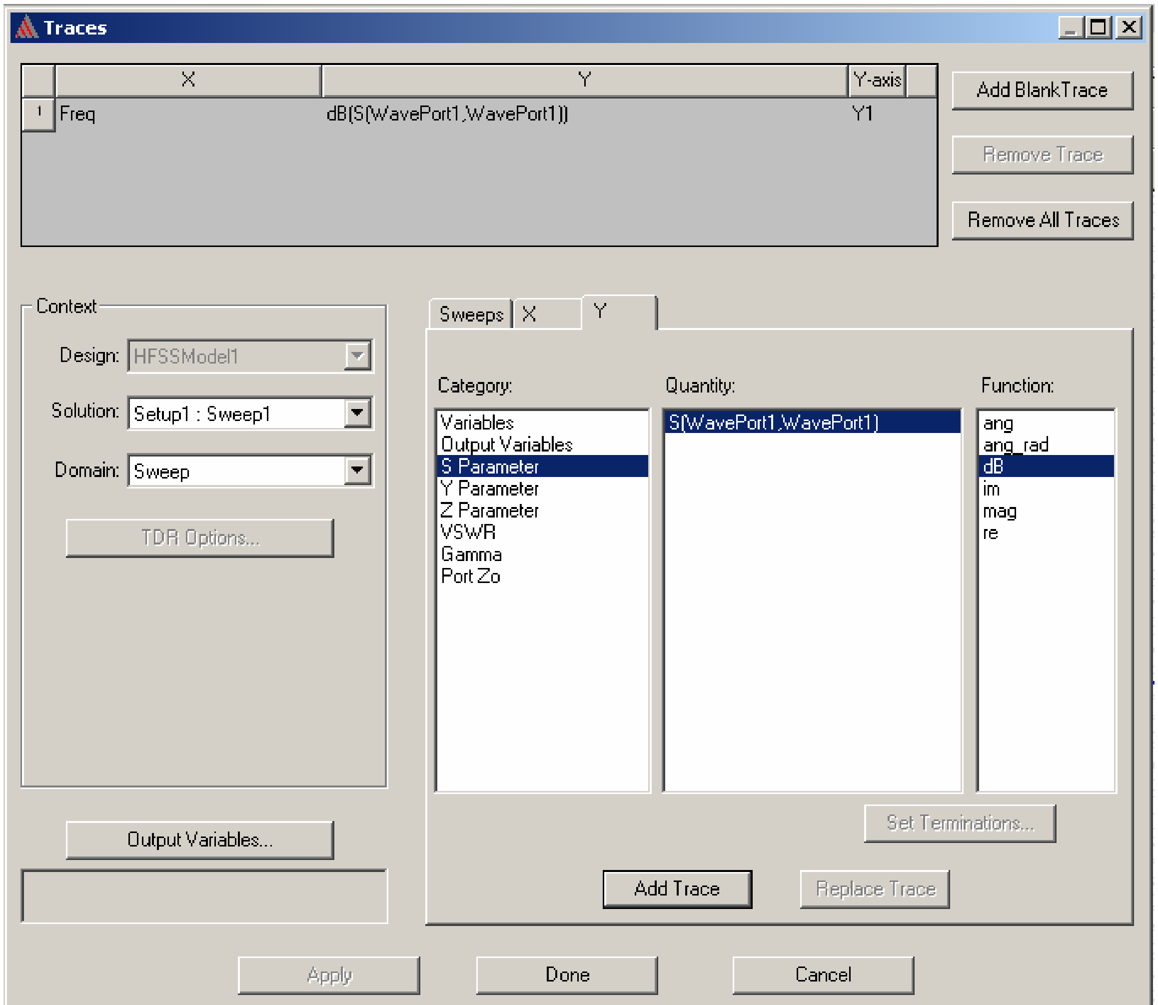

Figure 13: Adding traces window

Finally, the result should be a graph similar to the one in the following figure. It is clear that almost all of the energy is reflected at the input terminals of the empty cavity since there is no mechanism of energy loss within the cavity volume.

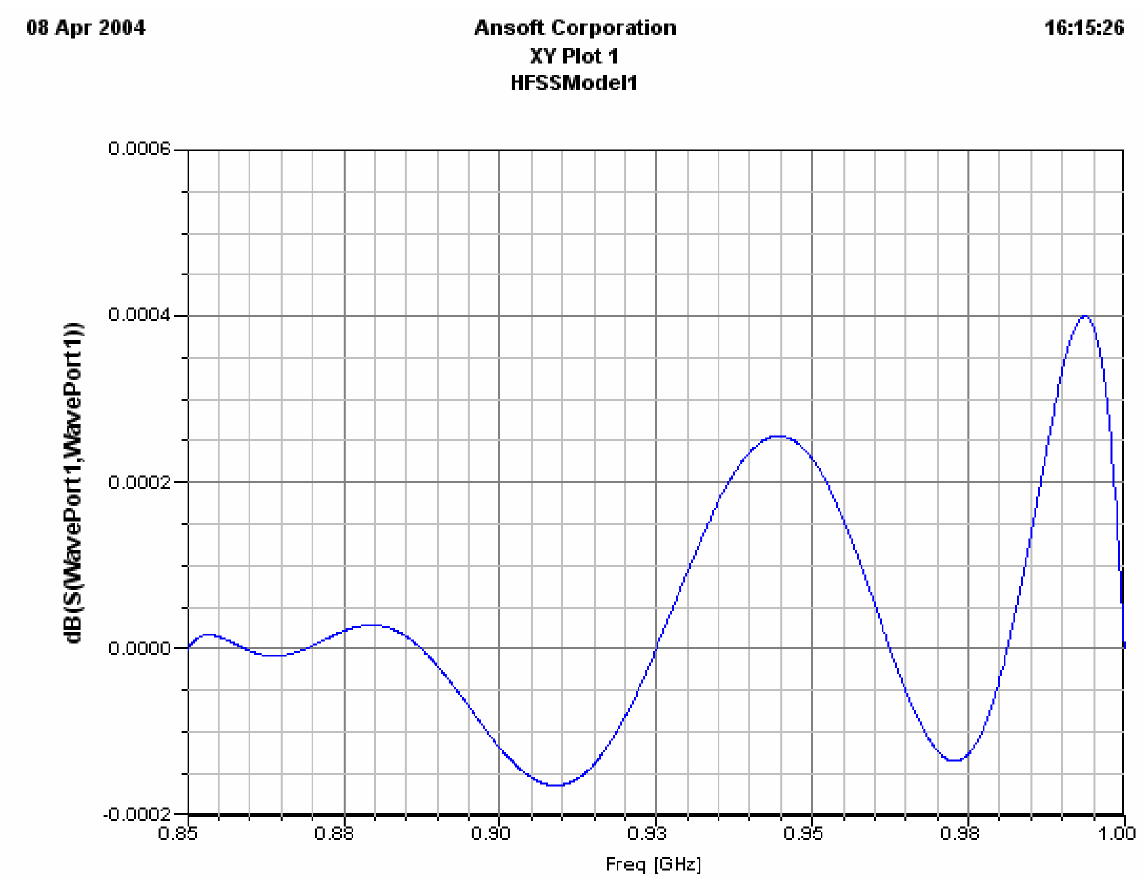

Figure 14 S_{11} versus frequency of the empty cavity

Next, a dielectric load with ε_r= 3-j0.1 is inserted into the cavity through the cutoff tube. The model is shown in Figure 15.

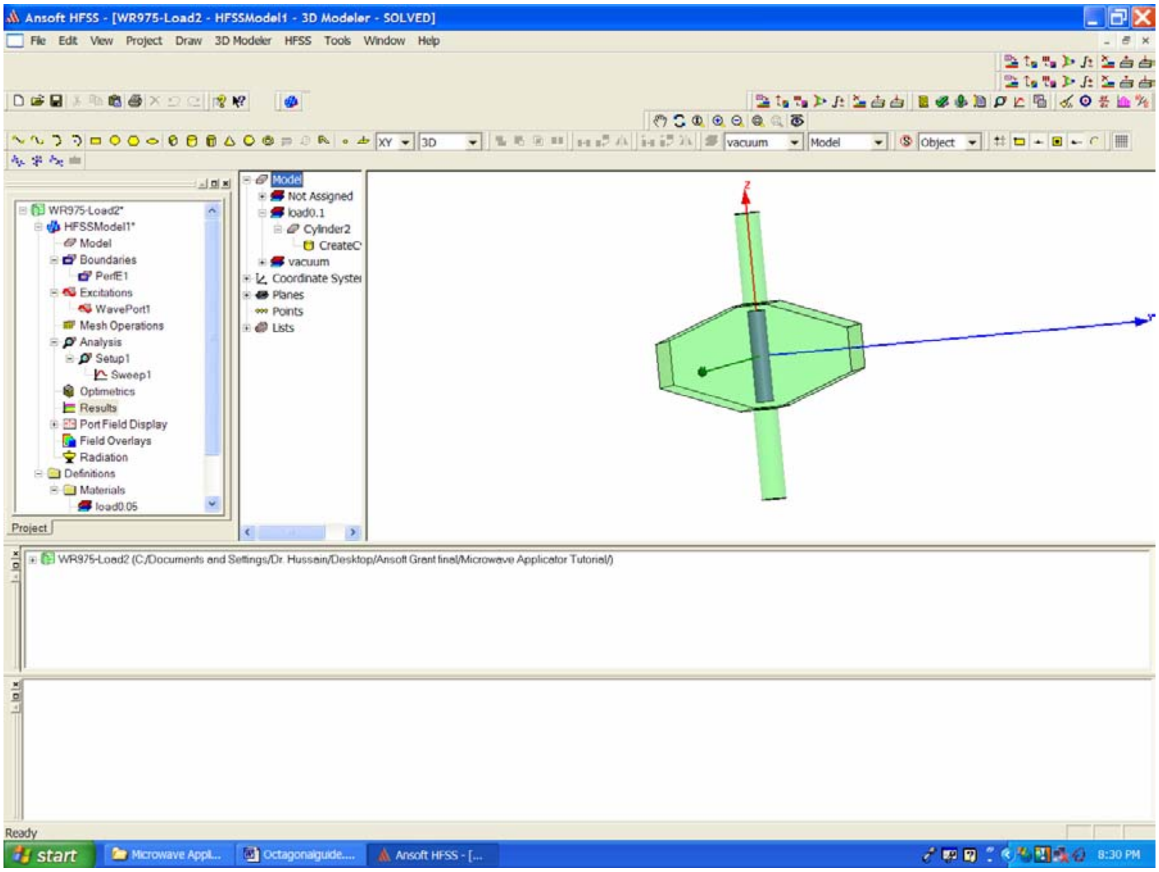

Figure 15 Geometry of the loaded cavity

The frequency spectrum of the reflection coefficient is shown in Figure 16 below:

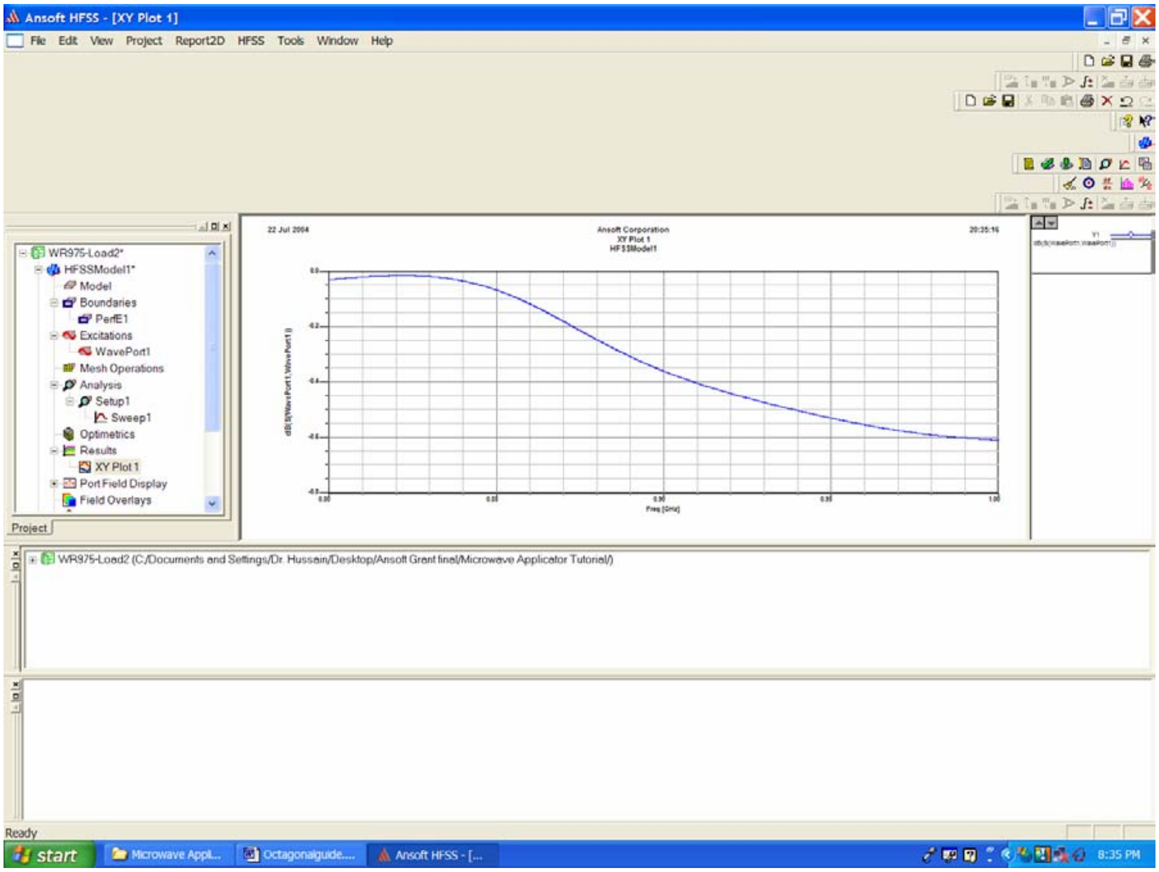

Figure 16 Frequency spectrum of the reflection coefficient of the loaded cavity, ε_r= 3-j0.1

It is clear that some energy has been coupled into the cavity and hence absorbed by the load.

Finally, a lossy sample with is loaded into the cavity. As expected, more energy is dissipated within the load. Figure 17 depicts the frequency characteristics of the reflection coefficient.

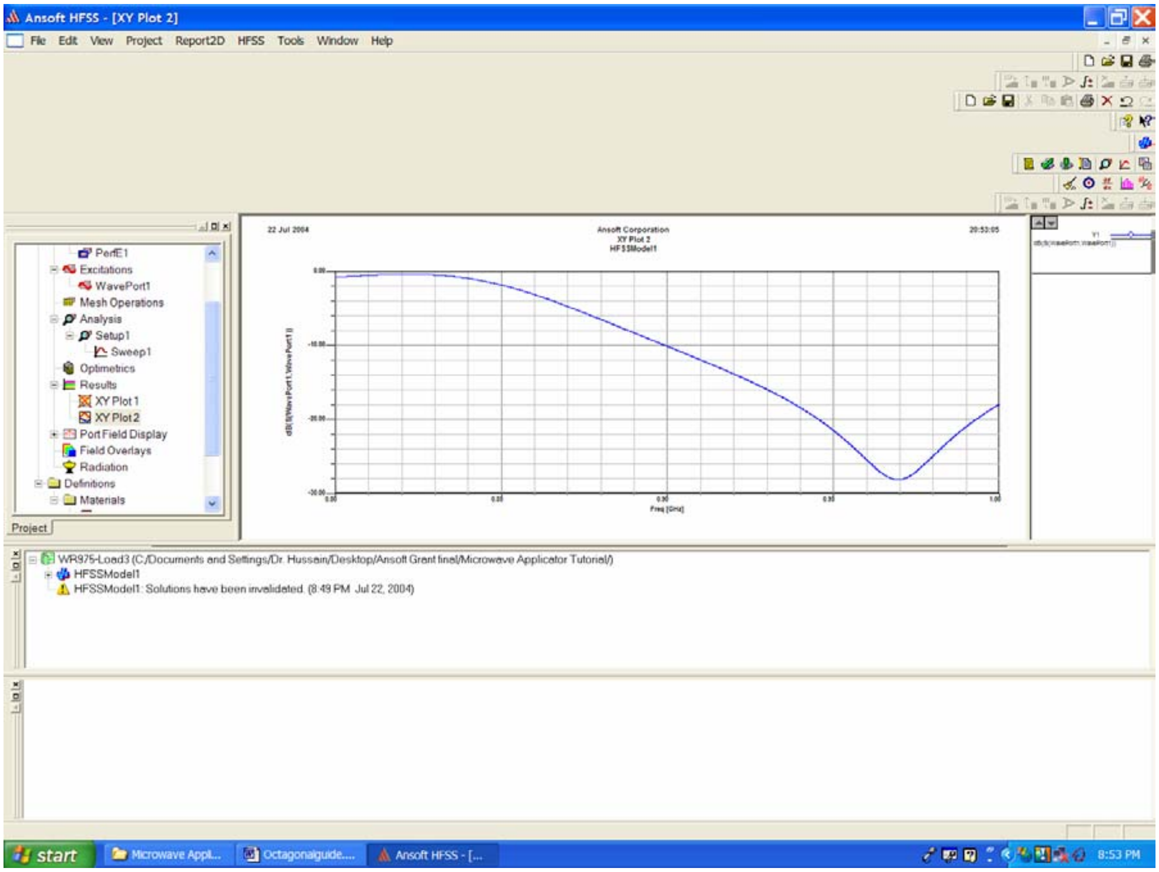

Figure 17 Frequency spectrum of the reflection coefficient of the loaded cavity, ε_r= 3-j3