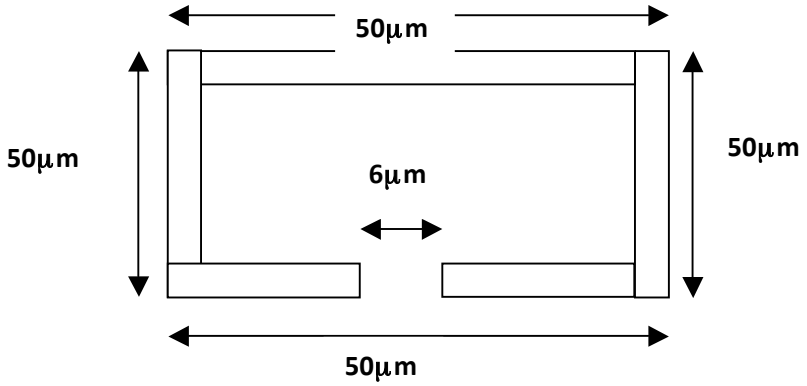
HFSS TUTORIAL

DESIGN OF A LOOP INDUCTOR

Design and simulate using HFSS the loop inductor with thickness of 2μm and metal width of 4μm. The inductor is located on an oxide layer with thickness of 8μm on a 10-Ω-cm silicon wafer of thickness 200μm. The conductivity of metal is 5.8×10^7 Siemens/m.



A) Calculate Q theoretically and plot Q using HFSS at 60 Ghz
B) Calculate L theoretically and plot L using HFSS at 60 Ghz
C) Calculate Cp for the model
D) Calculate Rp for the model
E) Calculate R series.

HFSS Software Setup

Follow these steps to set up your EE account for using the RF tools

1. Log on to apache server

NOTE: If you are working in ECSN 4.324, you can skip this section and go to number 2

Using SSH connect to apache.utdallas.edu server with your net ID username and password:

NOTE: You can use the apache server from just about any machine in ECSN Building, although it is recommended that you use the SunRay terminals in ECSN 4.324. If you use a Windows PC, make sure an *X-Window System Server* program is installed such as X-Win32 is installed and running, and that *Tunnel X11 connections* is enabled in your program.

2. Set up your environment

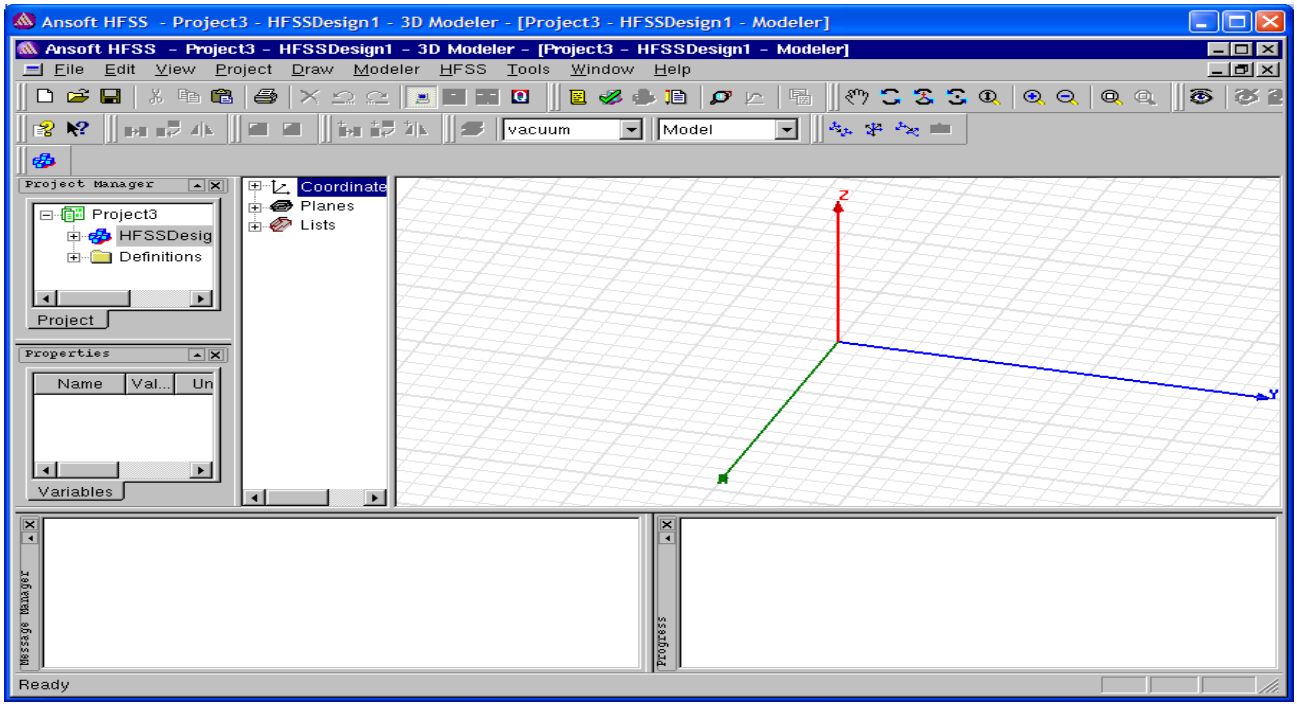
Set up your environment by entering following command.

```
. /proj/cad/startup/profile.hfss
```

Each time you open a shell to start HFSS, you need to enter this command.

Now type ***hfss*** on the terminal window to start HFSS

Once you have done this a new window with hfss interface will open as shown below



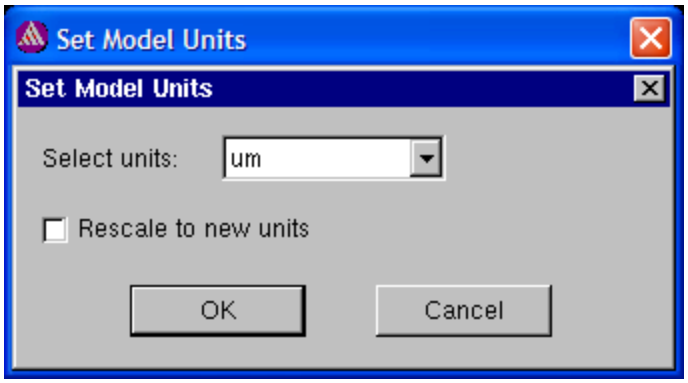
Opening and Renaming a New Project

To open a new project:

1. In an Ansoft HFSS window, select the menu item ***File - > New.***

To rename the project:

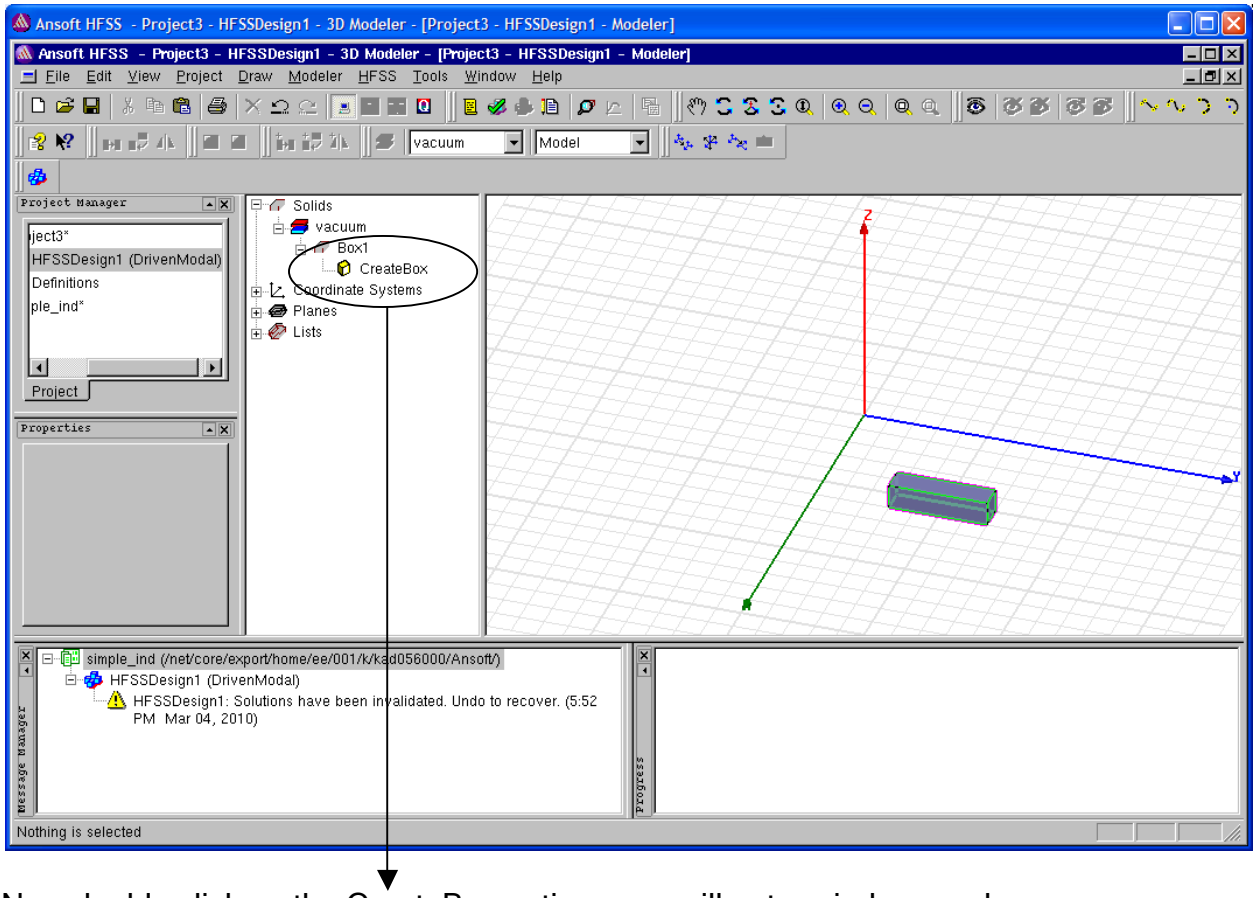
2. You can rename the project to a suitable name
   -To do this select the project in the ***project manager*** window and right click rename.
3. Select the Menu ***Modeler -> Units*** and change the units to **um**



Creating the inductor

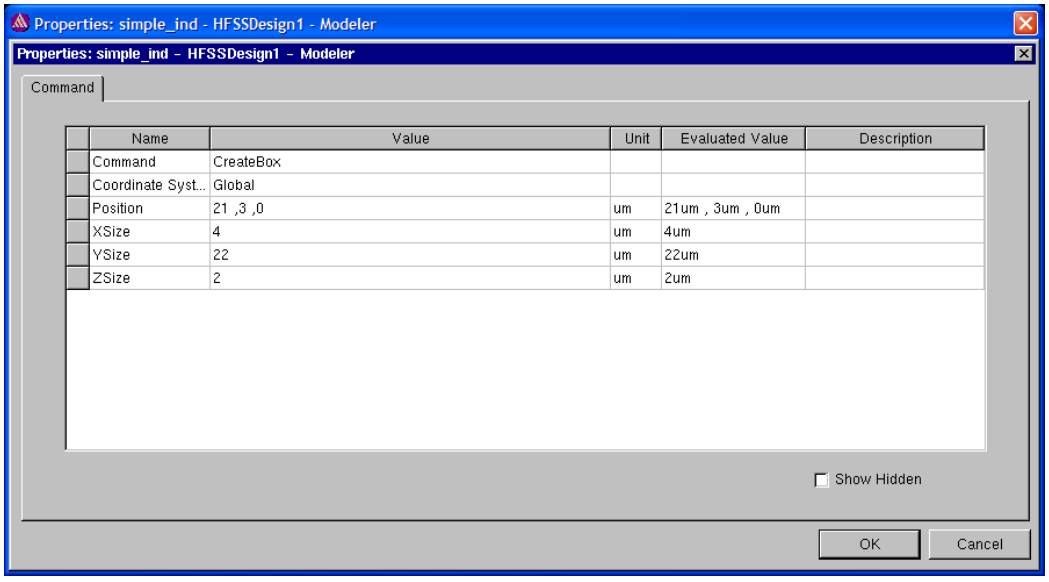
To create the inductor

1) Select the menu item Draw -> Box

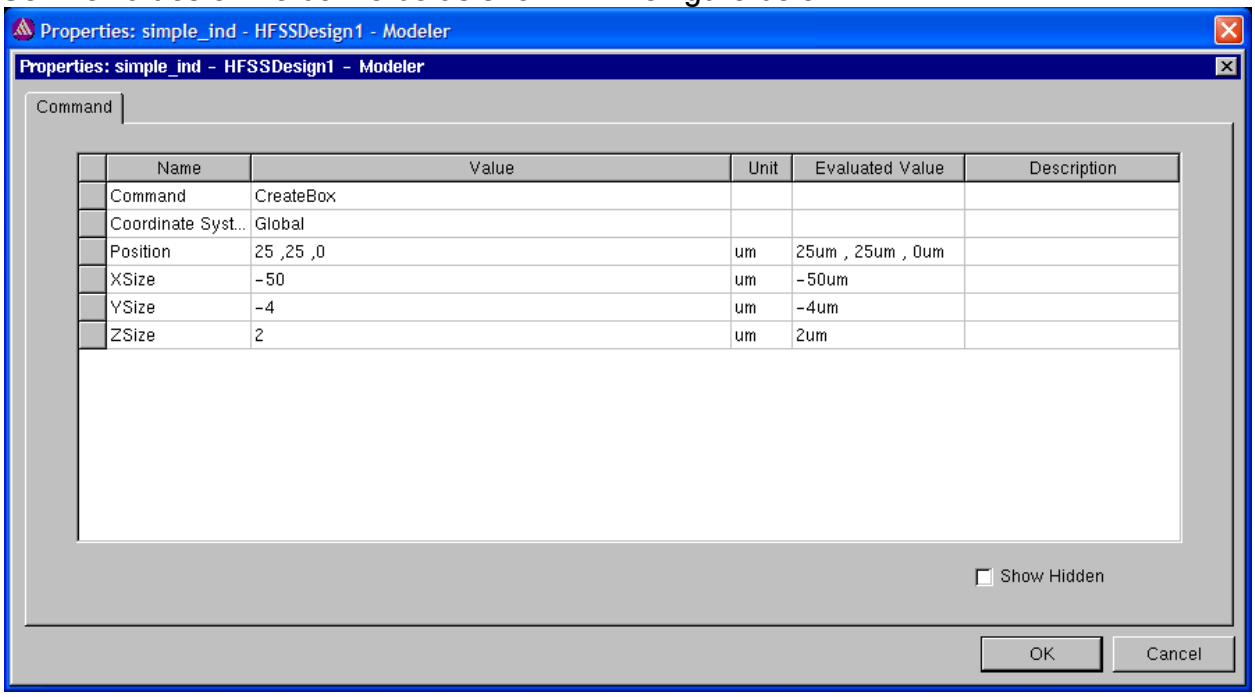


Now double click on the CreateBox options you will get a window as shown below.

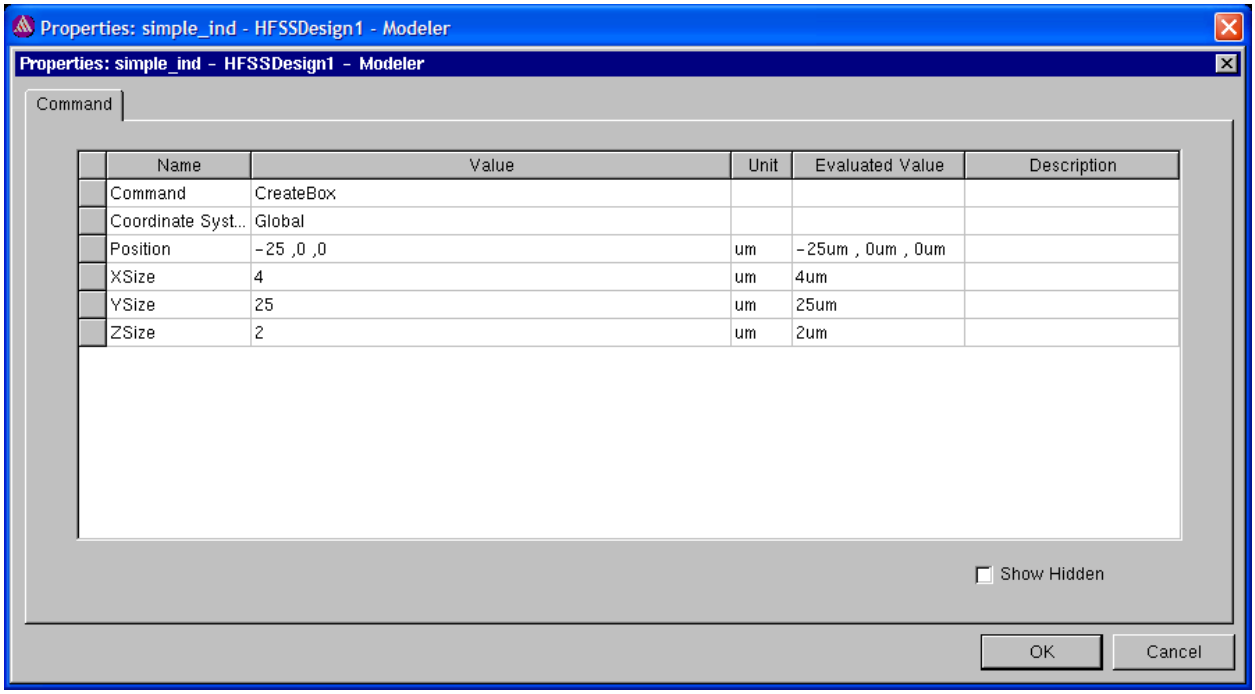
Set the values as shown in the figure



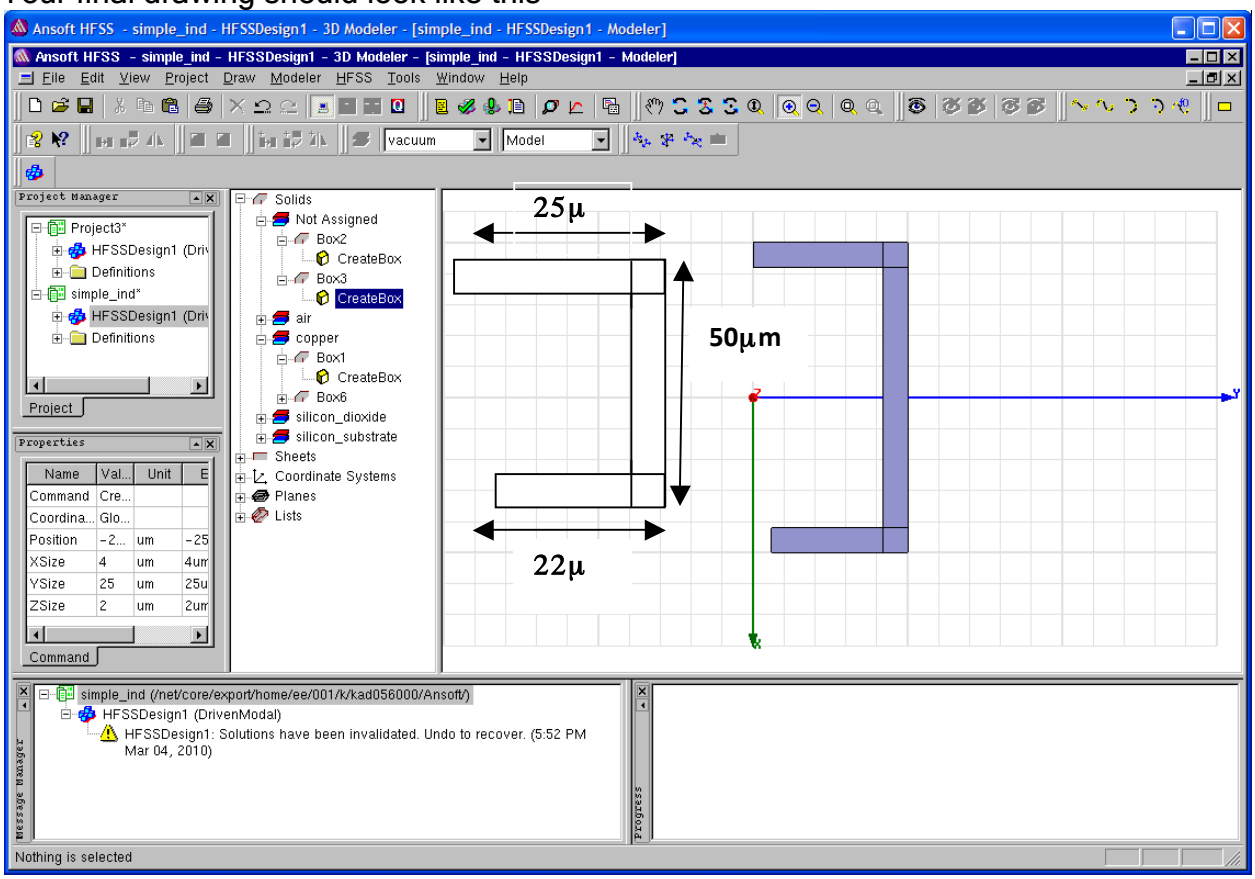
Repeat step 1 and draw a new box
Set the values of the box to be as shown in the figure below



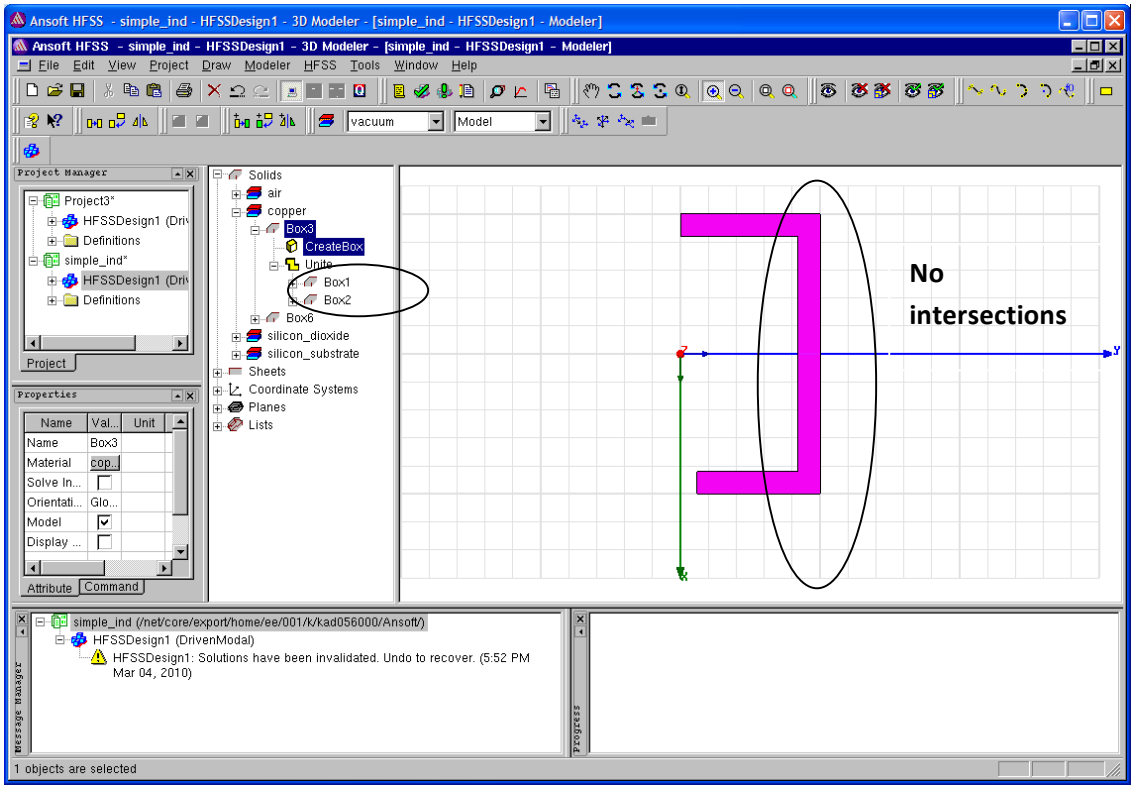
Repeat step 1 and draw a new box
Set the values of the box to be as shown in the figure below



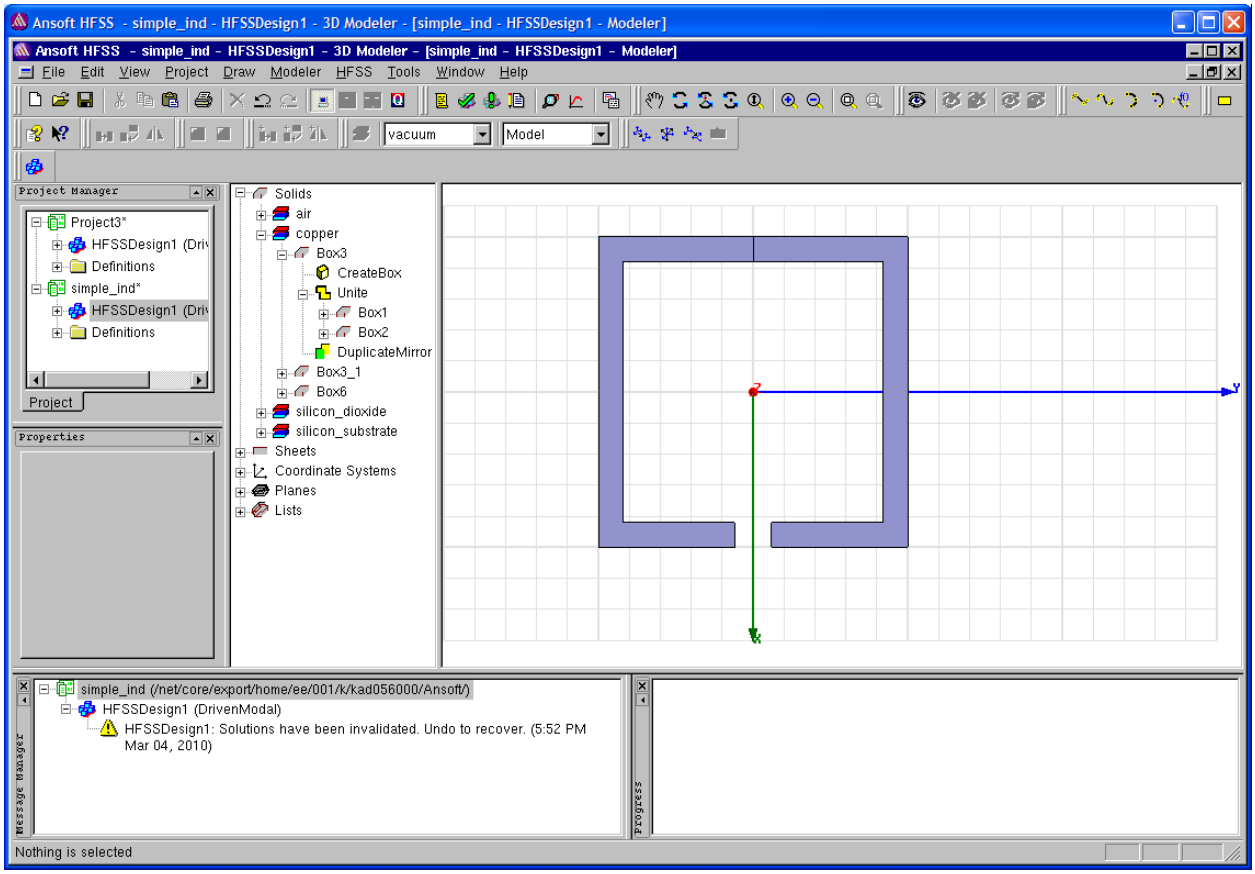
Your final drawing should look like this



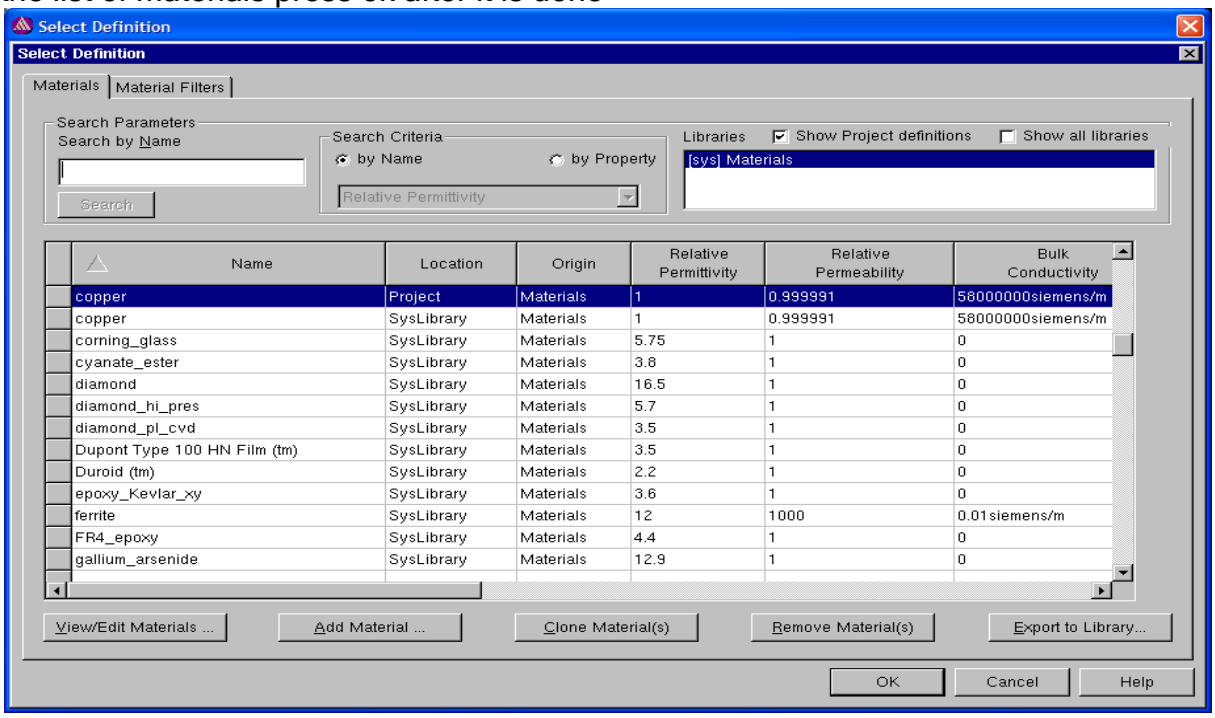
2) Select the three Boxes (Box1,2,3) by holding the CTRL key and then click the menu ***Modeler -> Boolean -> Unite*** .
   You will not see the intersection of all the boxes if the unite is done successfully and also a new **unite** button will be created as shown



3) Now you can mirror the same structure on the -Y direction
4) To mirror select the Box3 and select the menu ***Edit -> Duplicate -> Mirror***
5) The first click tells the anchor point to the tool so move the cursor to the origin of (X,Y,Z) plane and click it , the second click tells the tool the direction in which the mirror has to be done so move the cursor in -Y direction and then click again.
6) After doing this successfully you will see a DuplicateMirror button and the picture should look like this



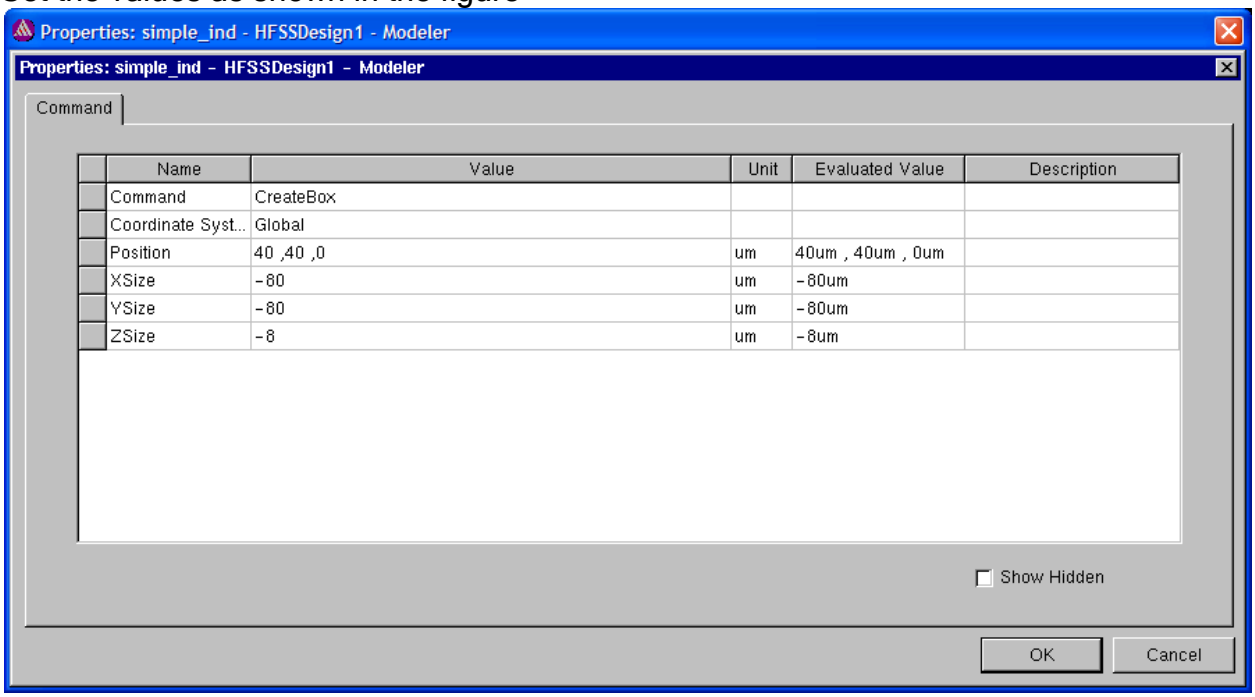
7) Repeat step 2 to unite the mirror structure with the original structure , the mirror structure will be name as the (Box3_1)
8) Select Box 3 and right click and select Assign materials now choose Copper from the list of materials press ok after it is done



Creating the Silicon-Oxide layer

To create Silicon-Oxide

1) Select the menu item Draw -> Box
   Now double click on the CreateBox options you will get a window as shown below.
   Set the values as shown in the figure

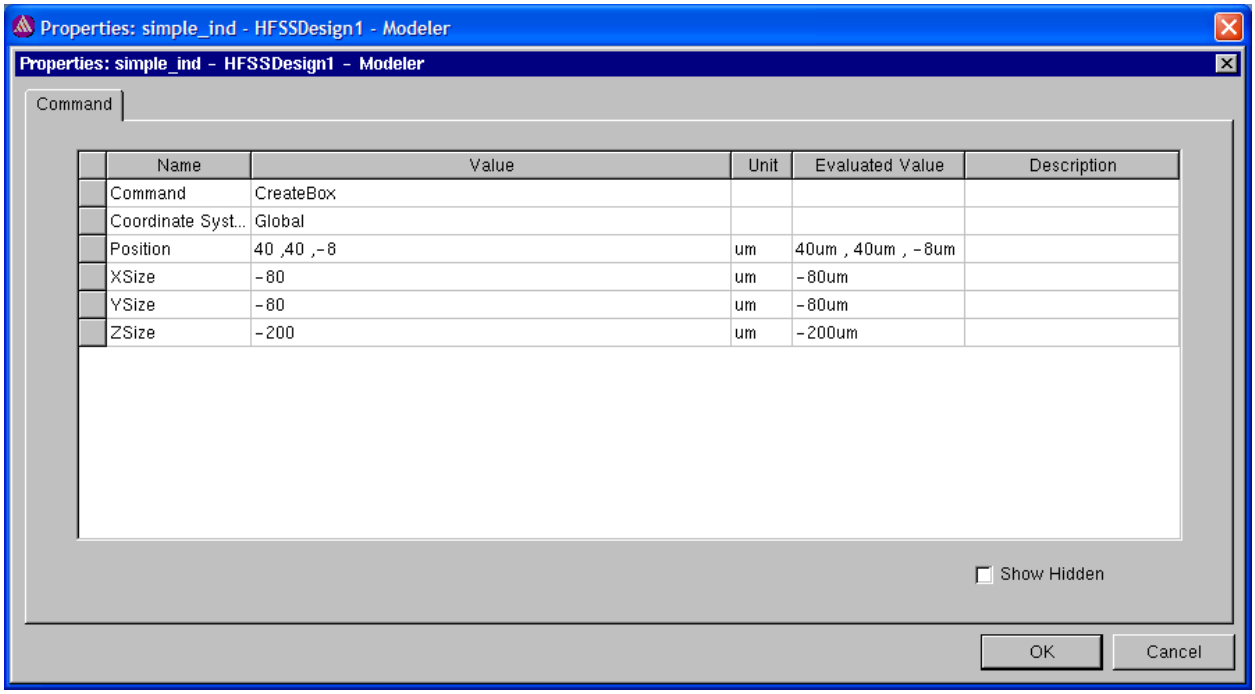


2) Select Box 4 and right click and select Assign materials now choose Silicon dioxide from the list of materials press ok after it is done

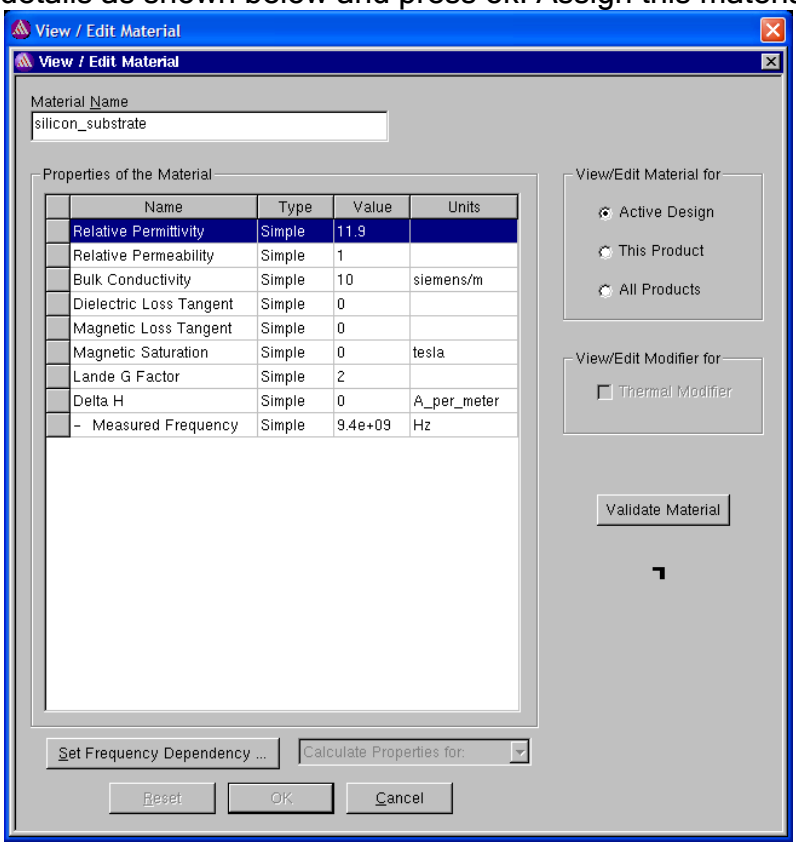
Creating the Silicon Substrate layer

To create Silicon-Substrate

1) Select the menu item Draw -> Box
   Now double click on the CreateBox options you will get a window as shown below.
   Set the values as shown in the figure



2) Select Box 5 and right click and select Assign materials. You will not see a silicon substrate material so you need to create a new material. Now click on the add material tab and in the Material Name type silicon_substrate and fill the other details as shown below and press ok. Assign this material to Box 5



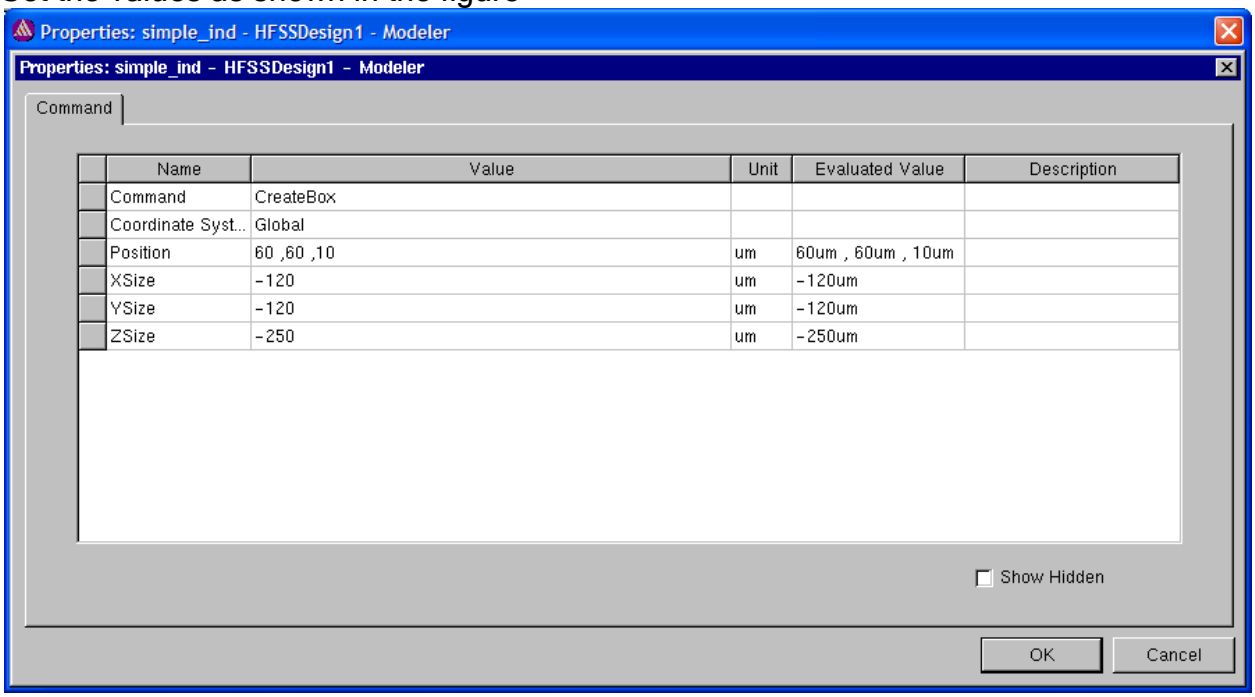
Relative permittivity = 11.9

Bulk Conductivity = 10

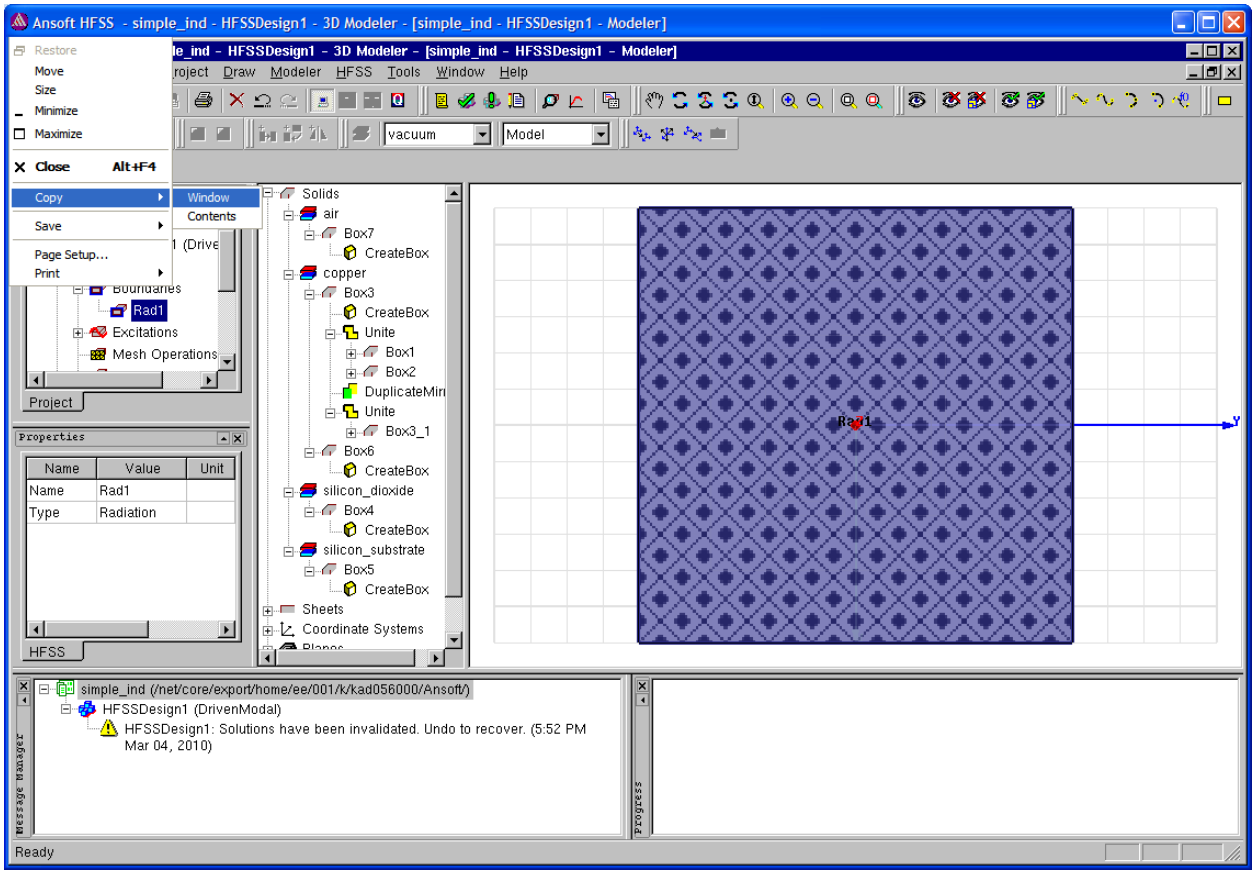
Creating the Air layer

To create Air layer

1) Select the menu item Draw -> Box
   Now double click on the CreateBox options you will get a window as shown below.
   Set the values as shown in the figure



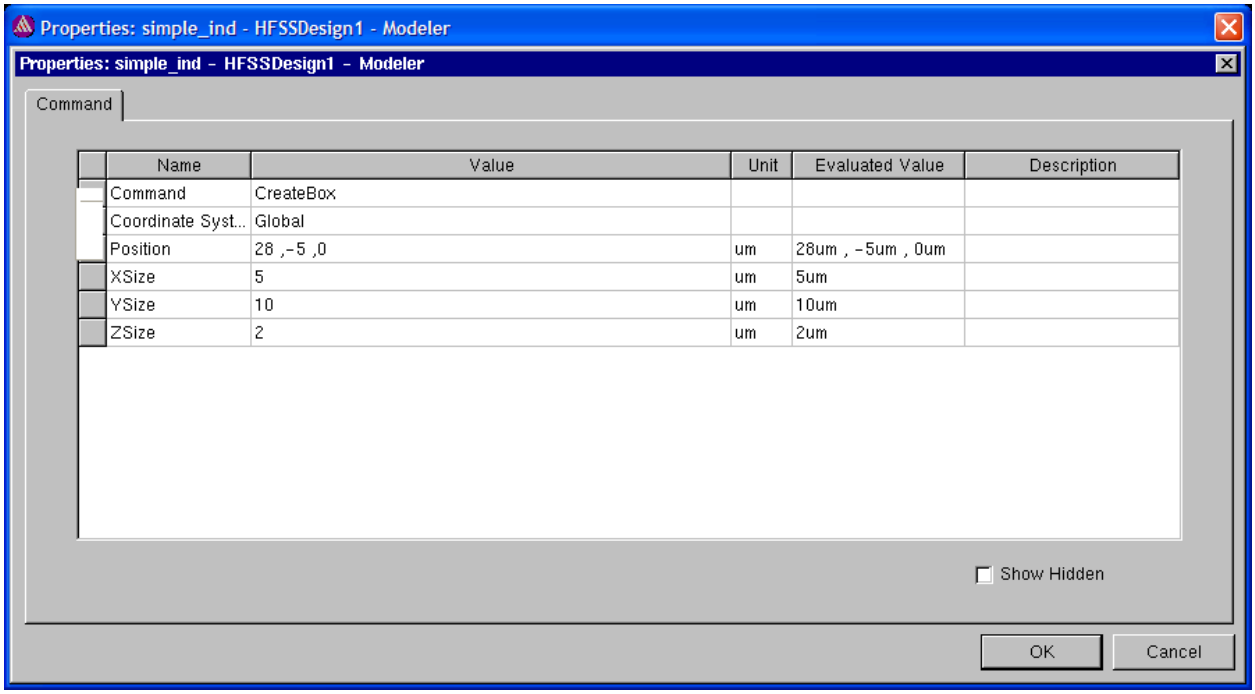
2) Select Box 6 and right click and select Assign materials. Select air and click ok
3) Now select Box 6 and right click and select Assign Boundary and click radiation you will see a new form press ok
4) You can verify that the boundary is assign from the project manager window select your ***project -> hfss design -> Boundary*** it should show **Rad1**
   After you click Rad1 in the project manager this is how the picture should look



Creating the GND layer

To create GND layer

1) Select the menu item Draw -> Box
   Now double click on the CreateBox options you will get a window as shown below.
   Set the values as shown in the figure

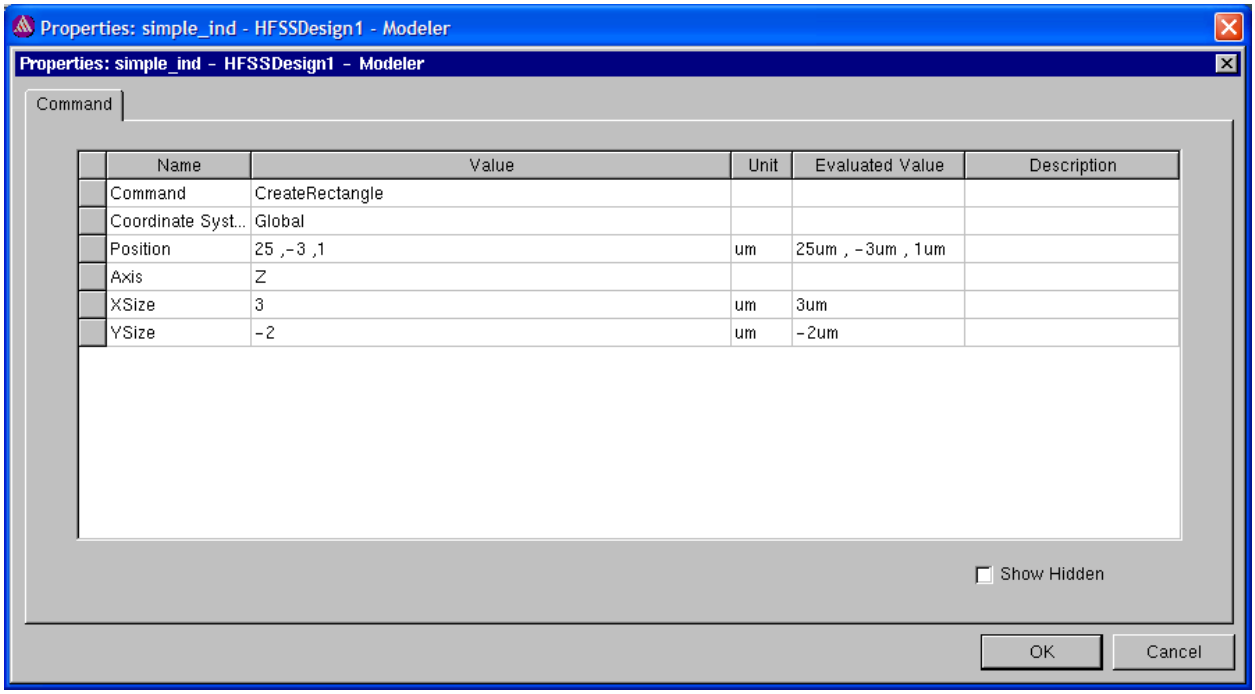


2) Select Box 7 and right click and select Assign materials. Select copper and click ok

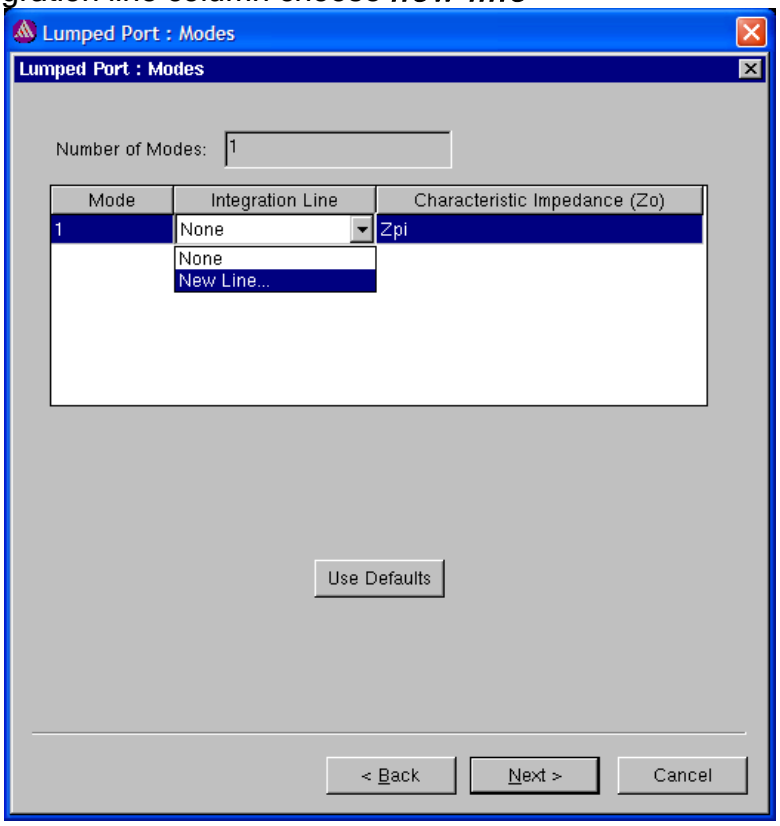
Creating the two port network

To create two port network

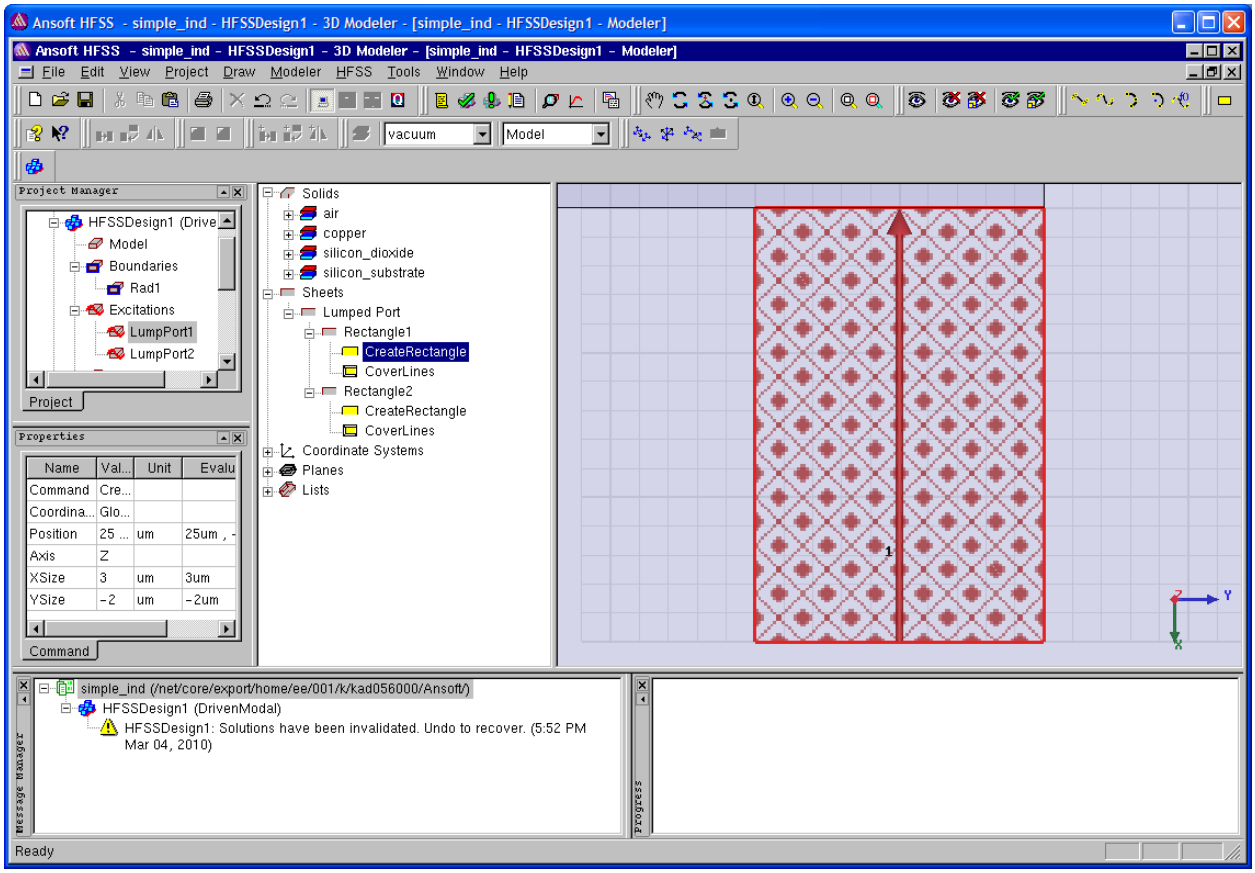
1) Select the menu item Draw -> rectangle
   Now double click on the CreateRectangle options you will get a window as shown below.
   Set the values as shown in the figure



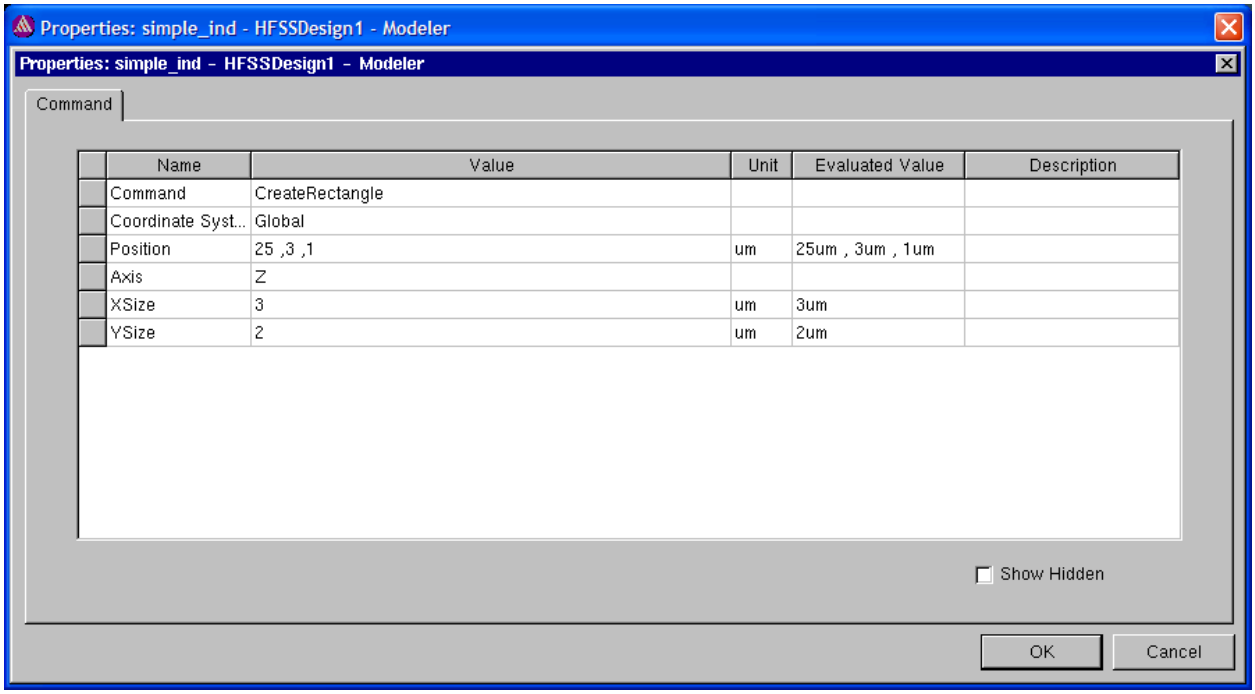
2) Select Rectangle 1 and zoom to the Rectangle 1 such that rectangle 1 fits in the window
3) Now select the Rectangle 1 and right click select assign excitation -> Lumped port you will see a new form assign Lump port1 as the name and click next
4) In the Integration line column choose ***new line***



5) As soon as you select the New line you will jump to the main window now move the cursor to the edge of the rectangle (towards gnd) you will see a small triangle click once on the triangle then move cursor to opposite side you will see a small triangle click again and you will see the same form you saw in step 4 but now the integration line column will have Defined as the value and click next and then finish. Your window should look like this



6) Repeat step1-5 for the next rectangle and assign the following values

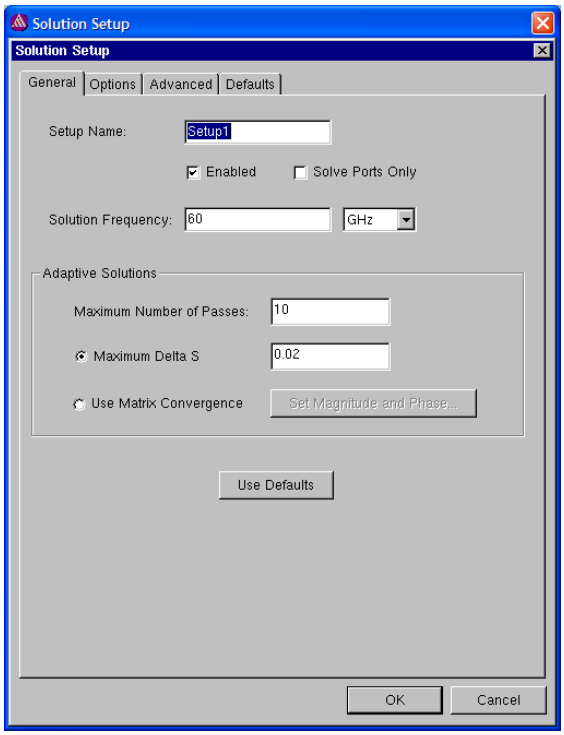


7) You can verify that the Lumped Port is assign from the project manager window select your ***project -> hfss design -> Excitation*** it should show **LumpPort1, LumpPort2**

Setting the Analysis

1) To set the analysis from the **project manager** window select your ***project -> hfss design -> analysis*** right click on analysis and select ***add solution setup***

Assign the following values and click ok

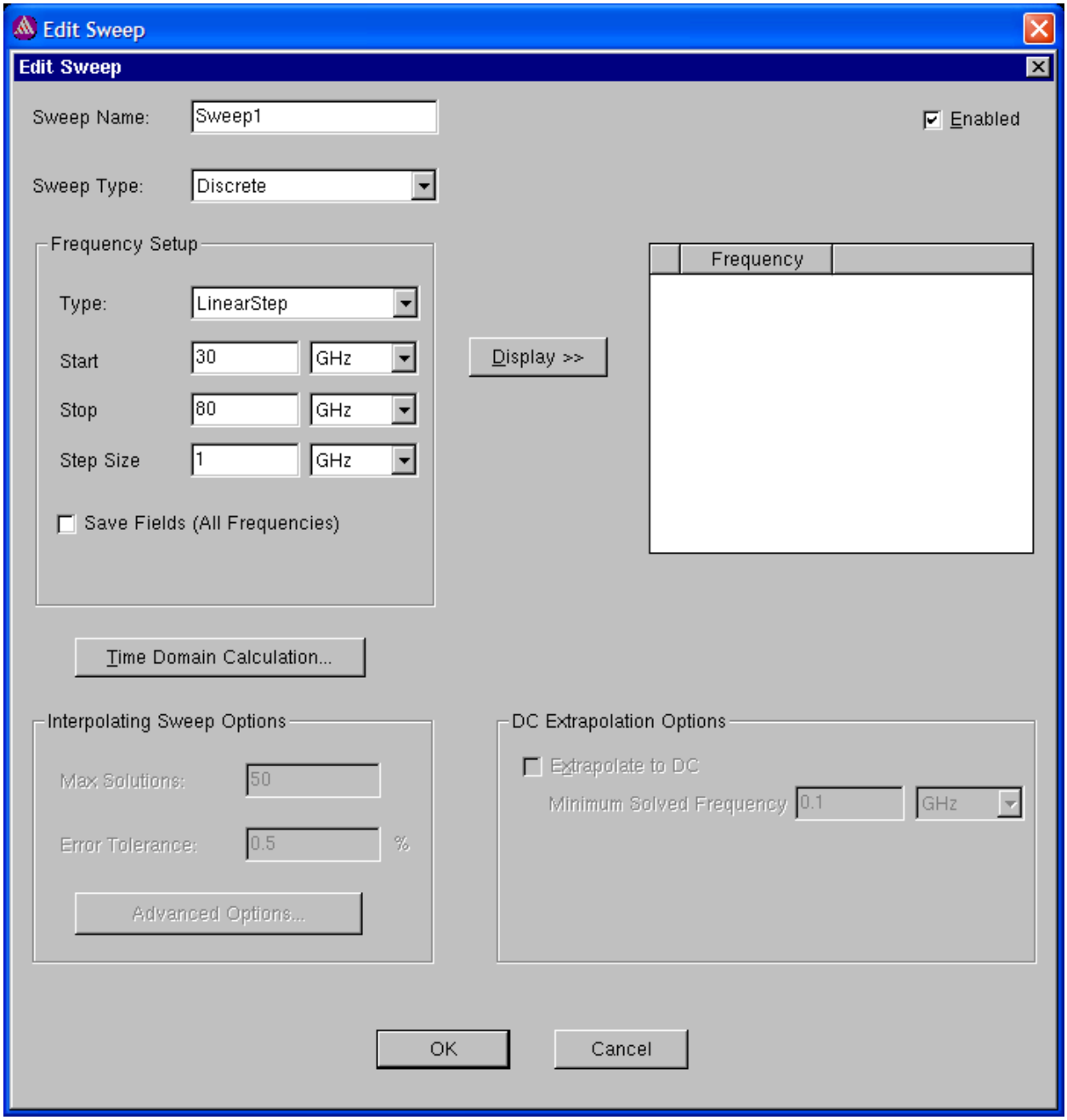


Solution Frequency = 60 Ghz

Maximum Number of Passes = 10

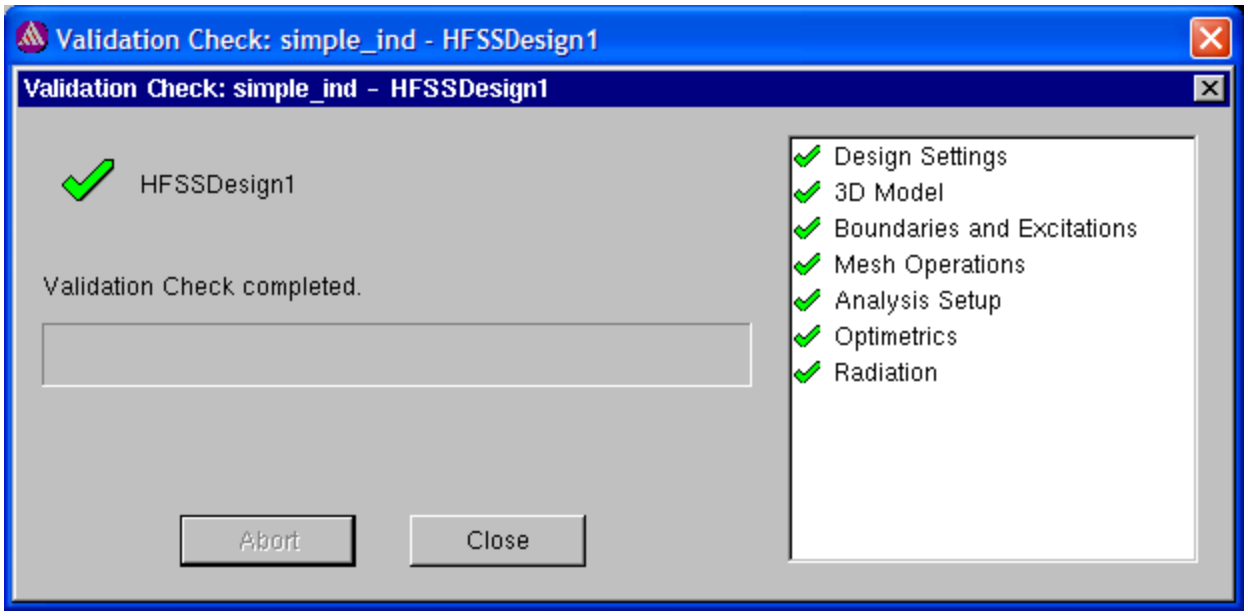
Maximum Delta S = 0.02

2) Right click on the analysis again and select ***add frequency sweep*** you will see the form shown below assign the values and press ok



Checking and Running the Design

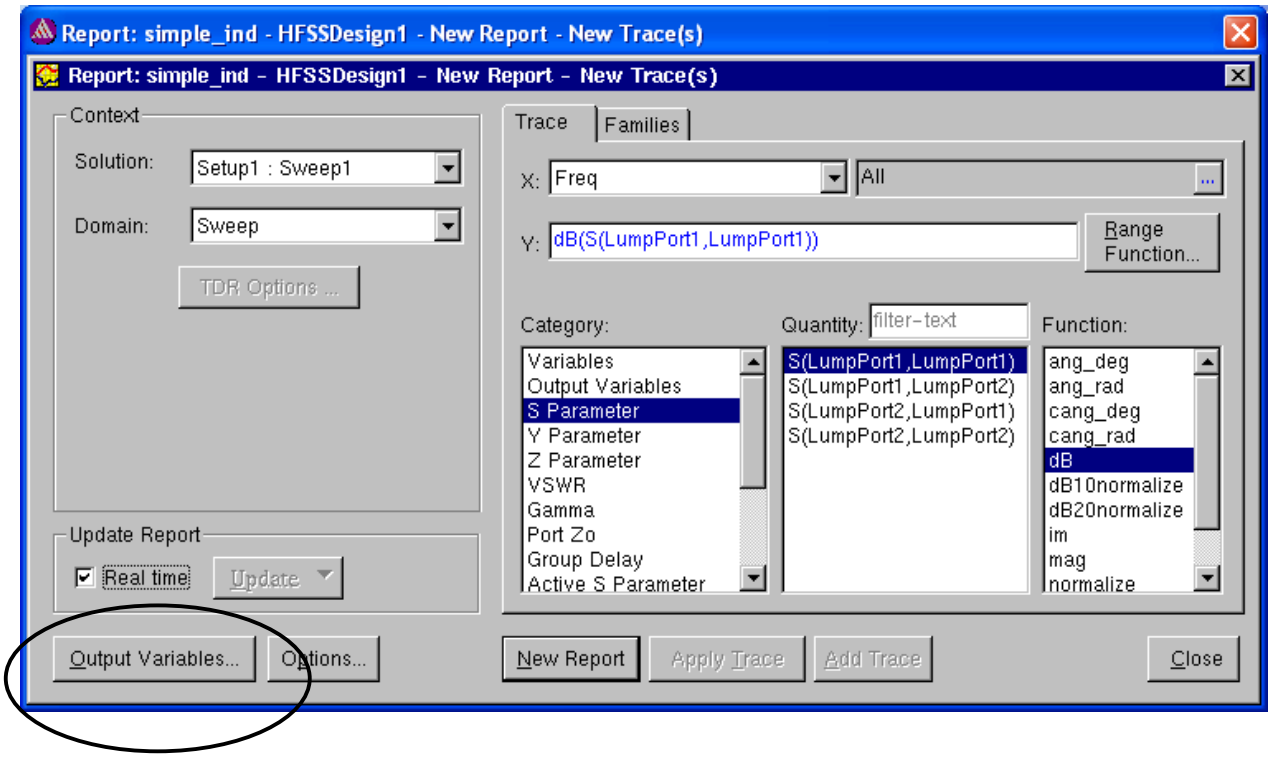
1) Once you have done the layout of inductor you need to check the design for any errors so select the Menu ***HFSS -> Validation*** check and you should get the following figure if everything is right. If there is a red cross across any of them that means that part is not done properly



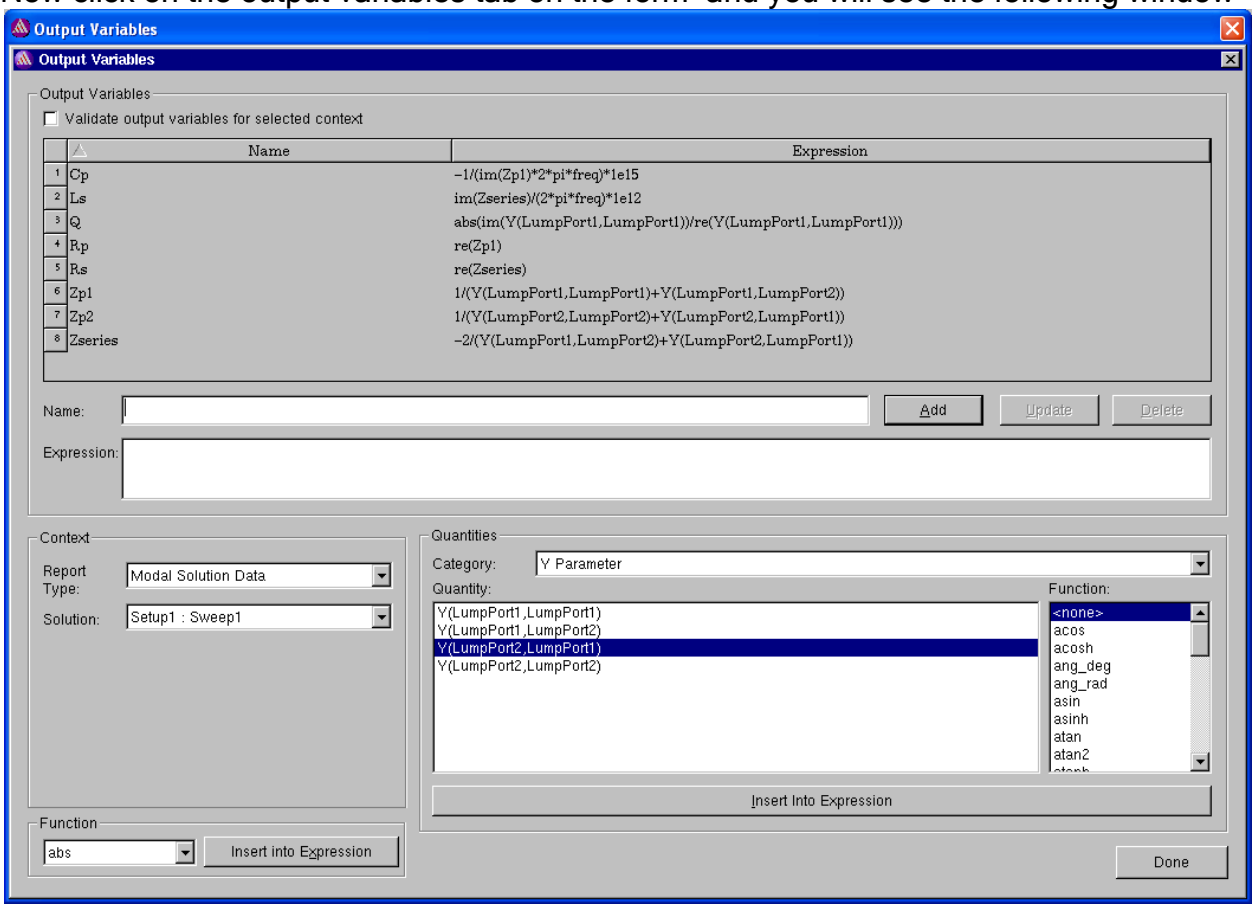
2) After this done close the dialog box and then select the menu ***HFSS -> Analyze all*** to run the simulation

Checking the results

After the simulation is done need to check that the simulation is run in normal completion at the bottom left of the window.
Now right click on the results tab in the Project manager window and select Create Modal solution Data report and then select rectangular plot, you should get a new window as shown below



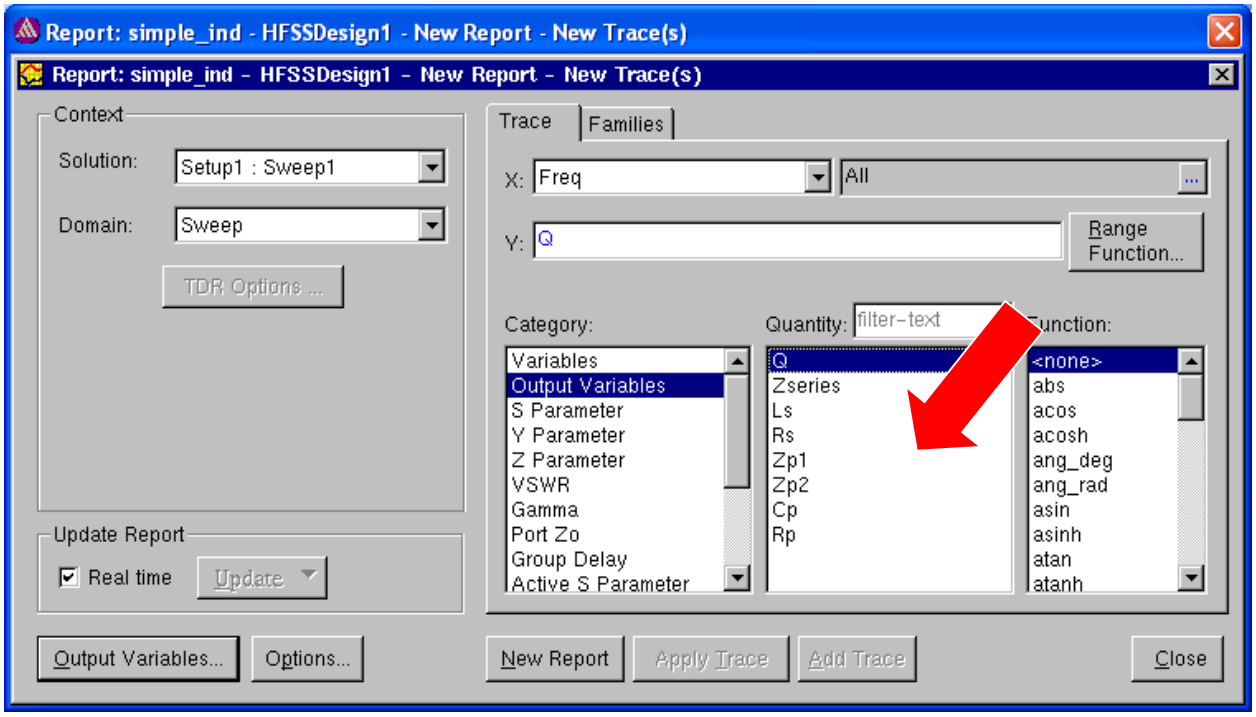
Now click on the output variables tab on the form and you will see the following window



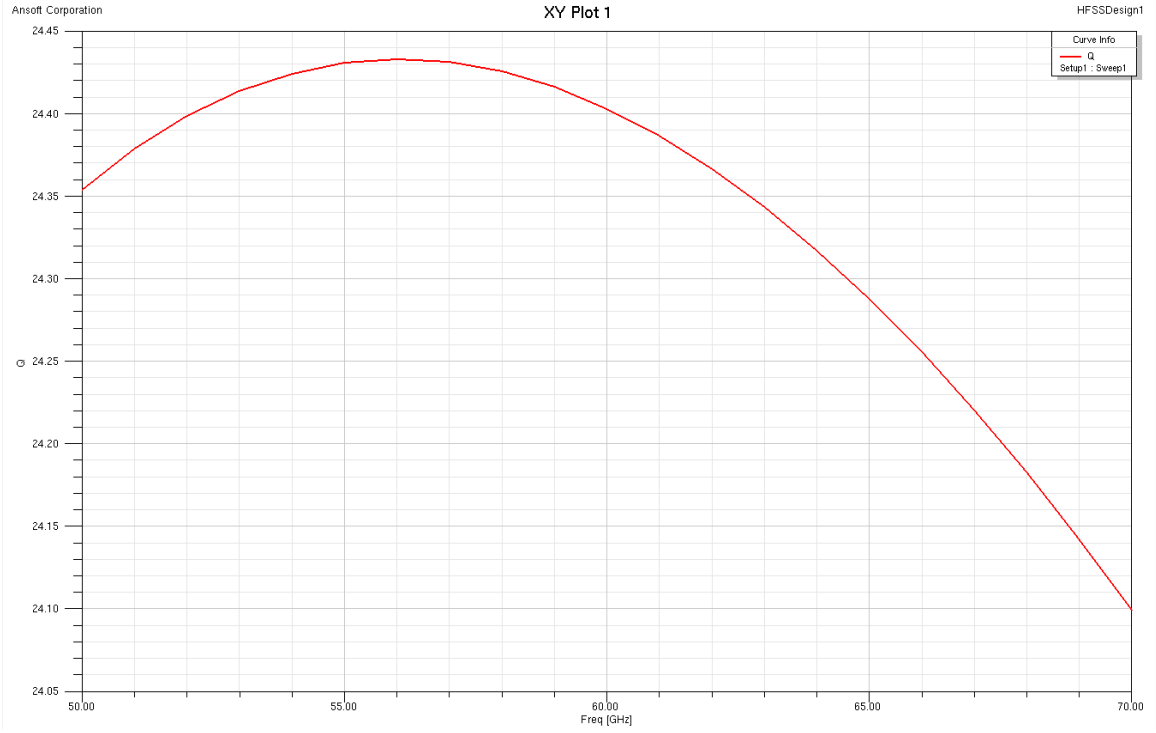
Insert the following in the form, to do so insert Zseries as name and add the expression in the expression tab and click add, after you finish entering all the terms the form should look like shown above then press done

Zseries = -2/(Y(LumpPort1,LumpPort2)+Y(LumpPort2,LumpPort1))
Zp1 = 1/(Y(LumpPort1,LumpPort1)+Y(LumpPort1,LumpPort2))
Zp2 = 1/(Y(LumpPort2,LumpPort2)+Y(LumpPort2,LumpPort1))
Q = abs(im(Y(LumpPort1,LumpPort1))/re(Y(LumpPort1,LumpPort1)))
Ls = im(Zseries)/(2*pi*freq)*1e12
Rs = re(Zseries)
Cp = -1/(im(Zp1)*2*pi*freq)*1e15
Rp = re(Zp1)

Now you will see all the values in the Output Variables tab as shown below

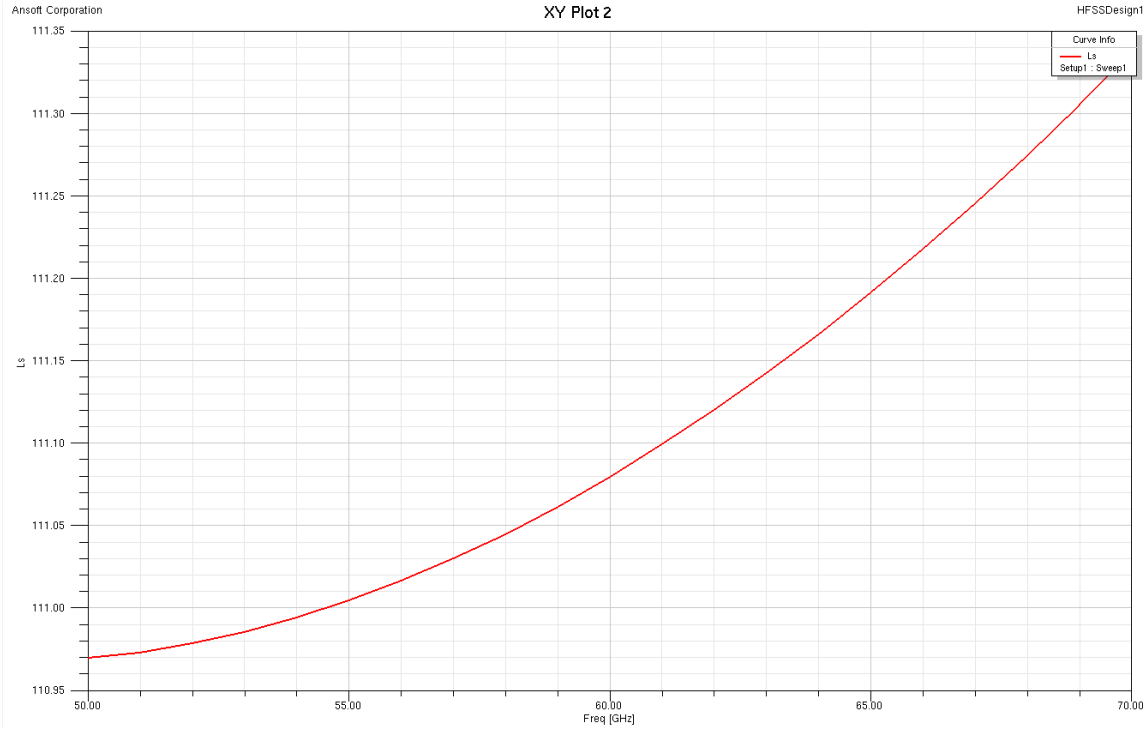


To see the result select the value you want to plot and hit New report
You should get the following results

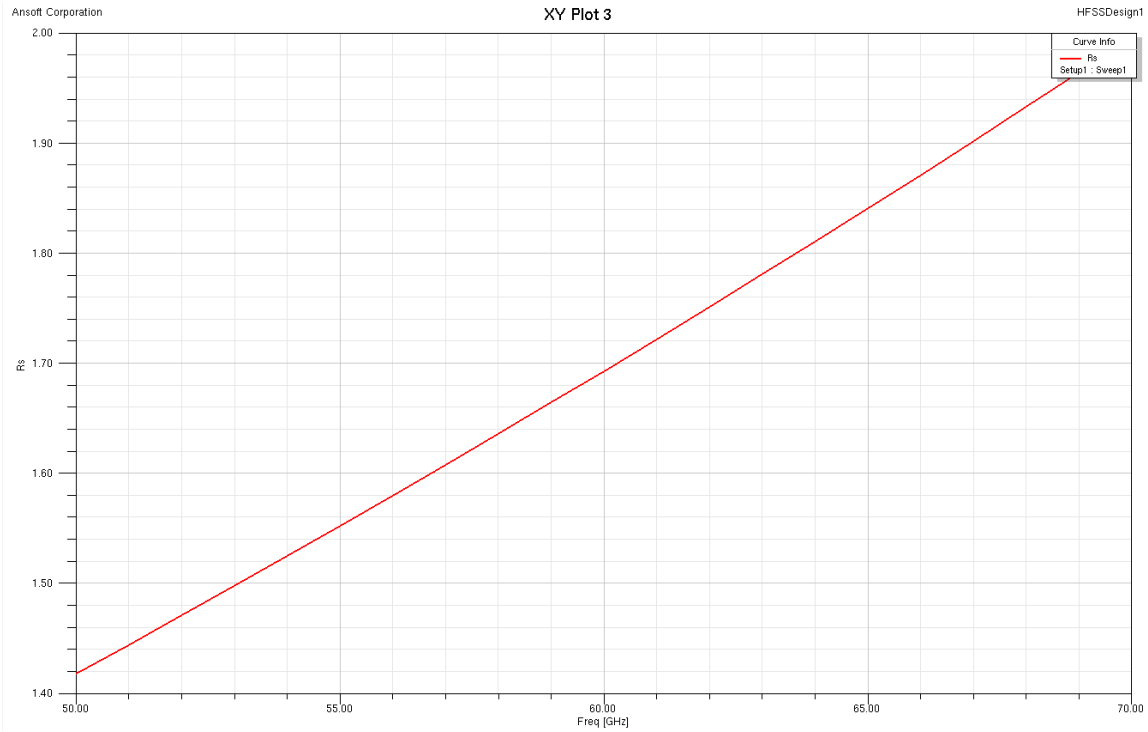


1) Plot of Q

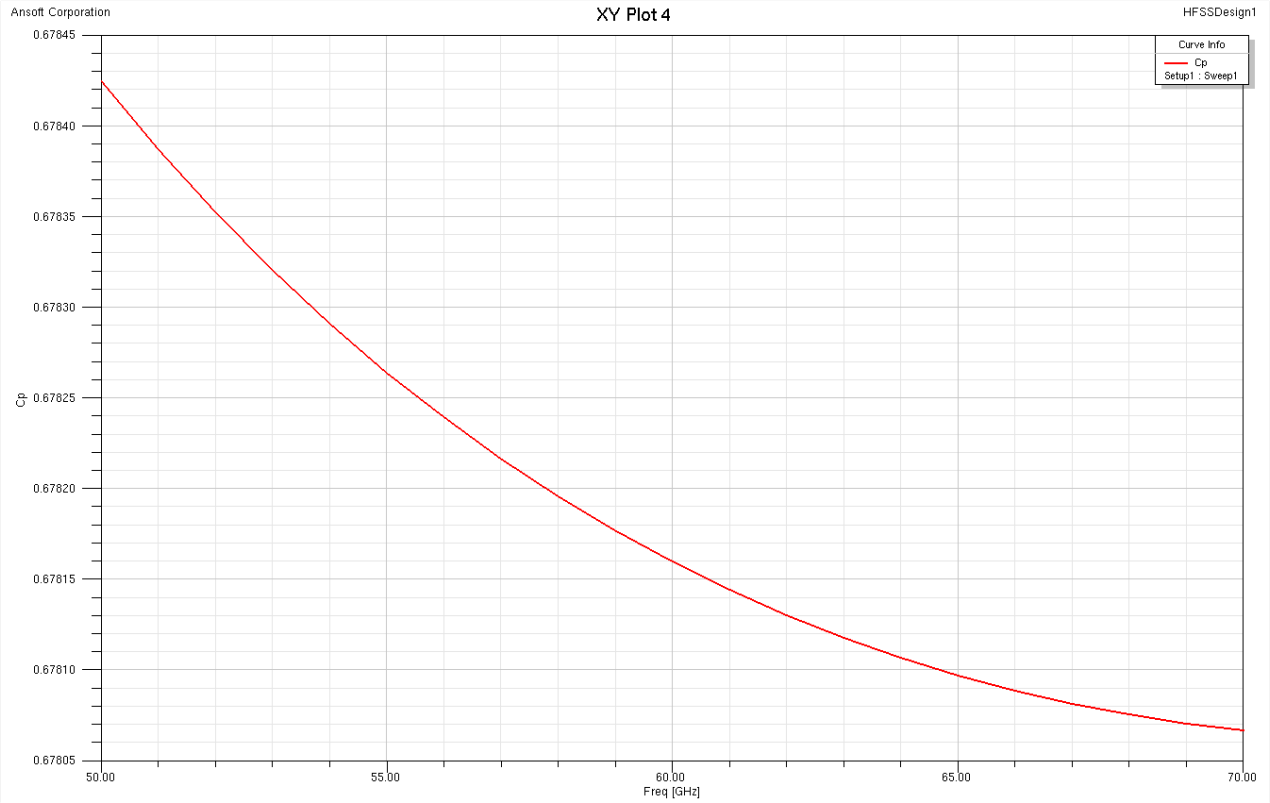
2) Plot of Ls in pH



3) Plot of Rs in Ω



4) Plot of Cp in fF



5) Plot of Rp in Ω