

# ADS Software

When we open the ADS icon or opened the software, the interface looks like as shown in Figure 1

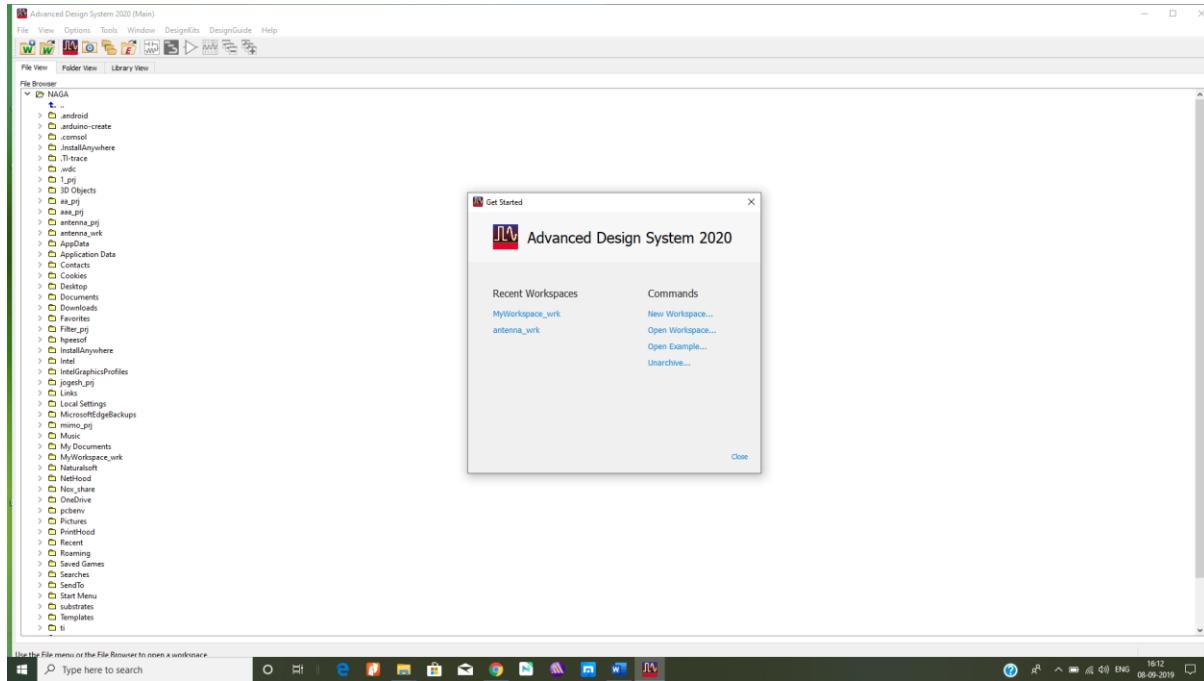
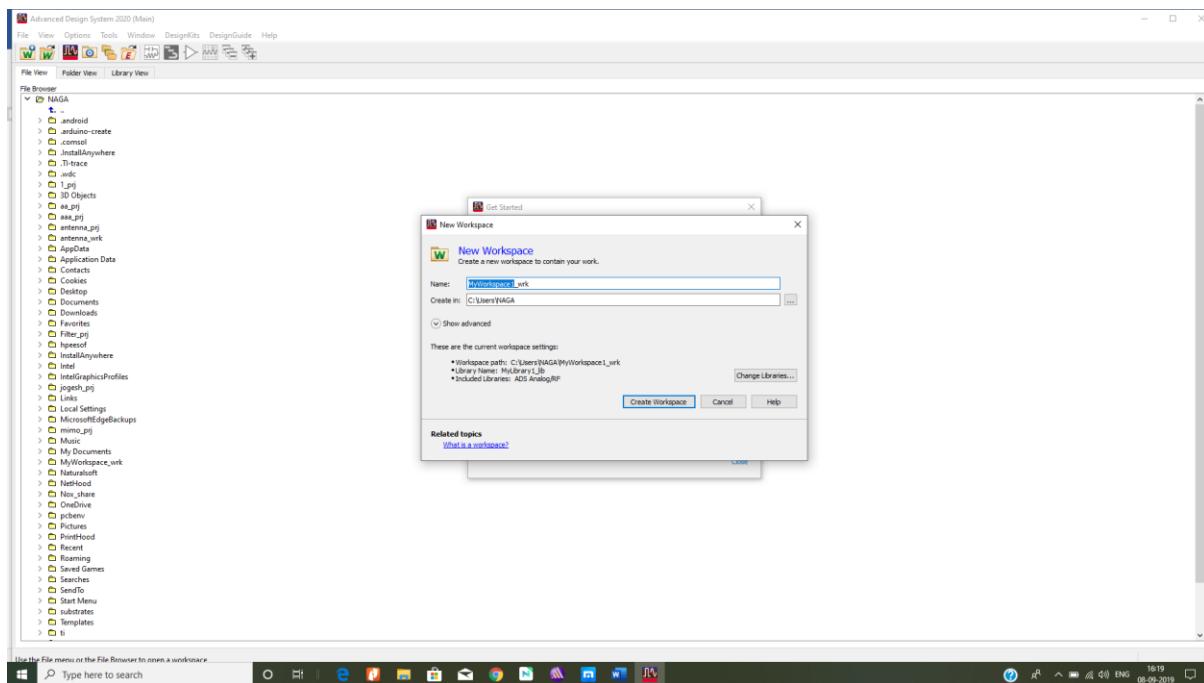
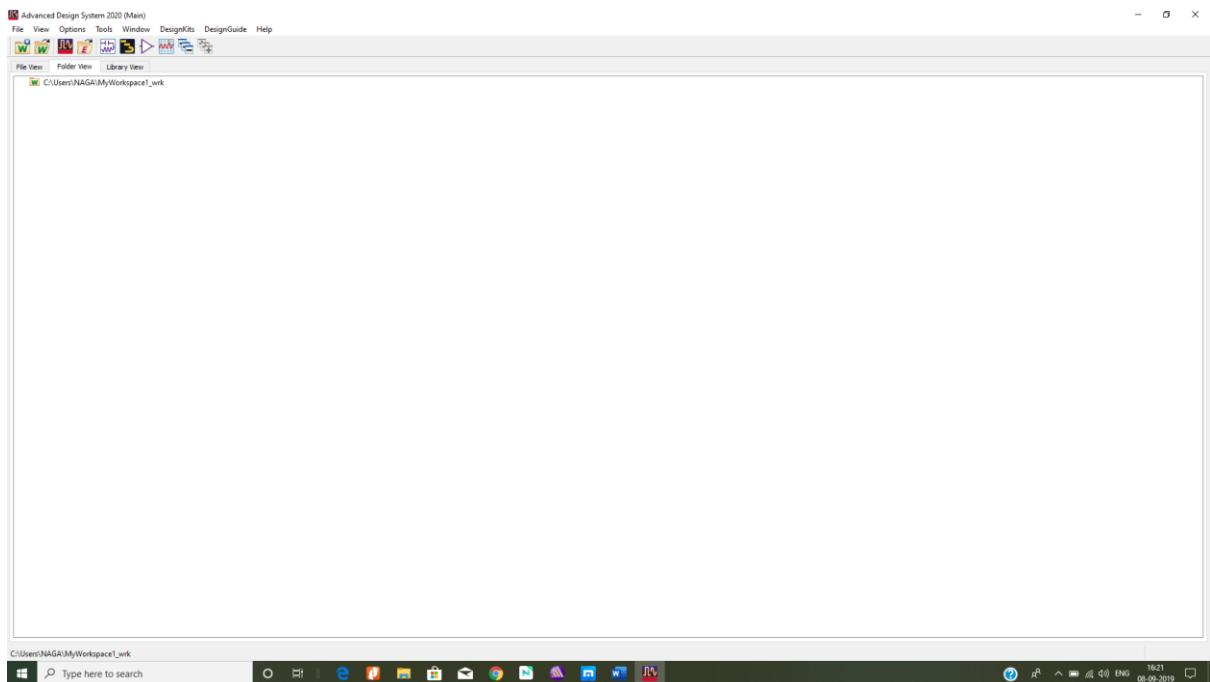


Figure 1

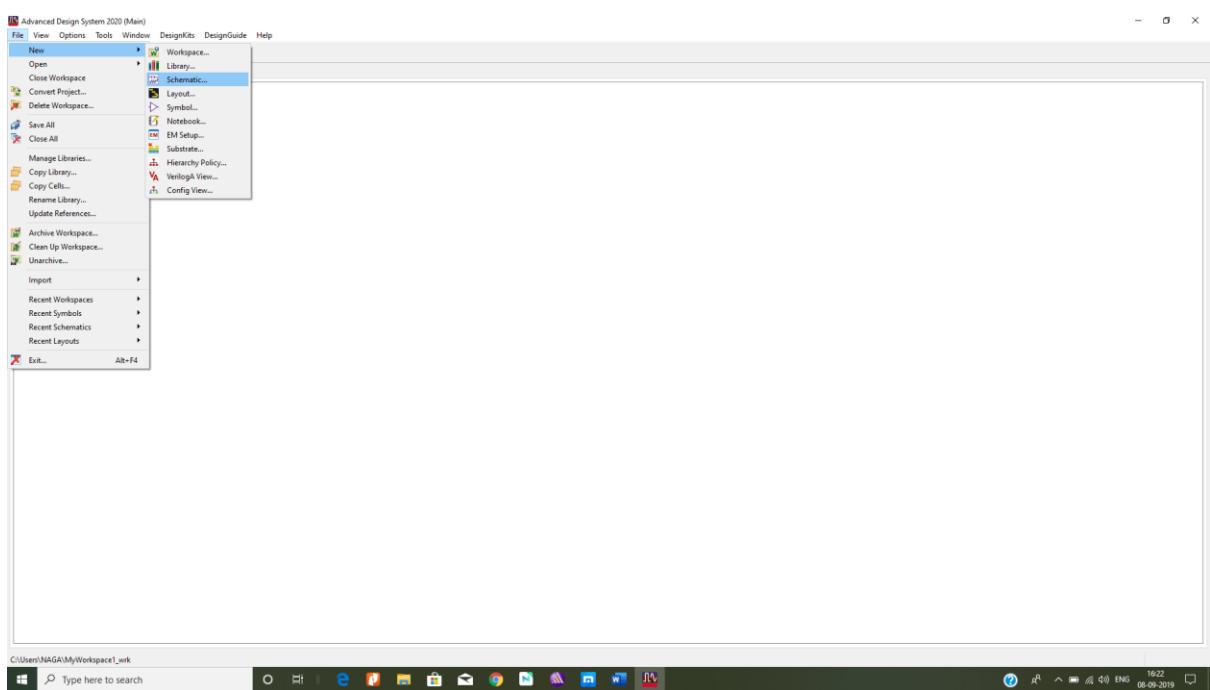
Click on New Workspace



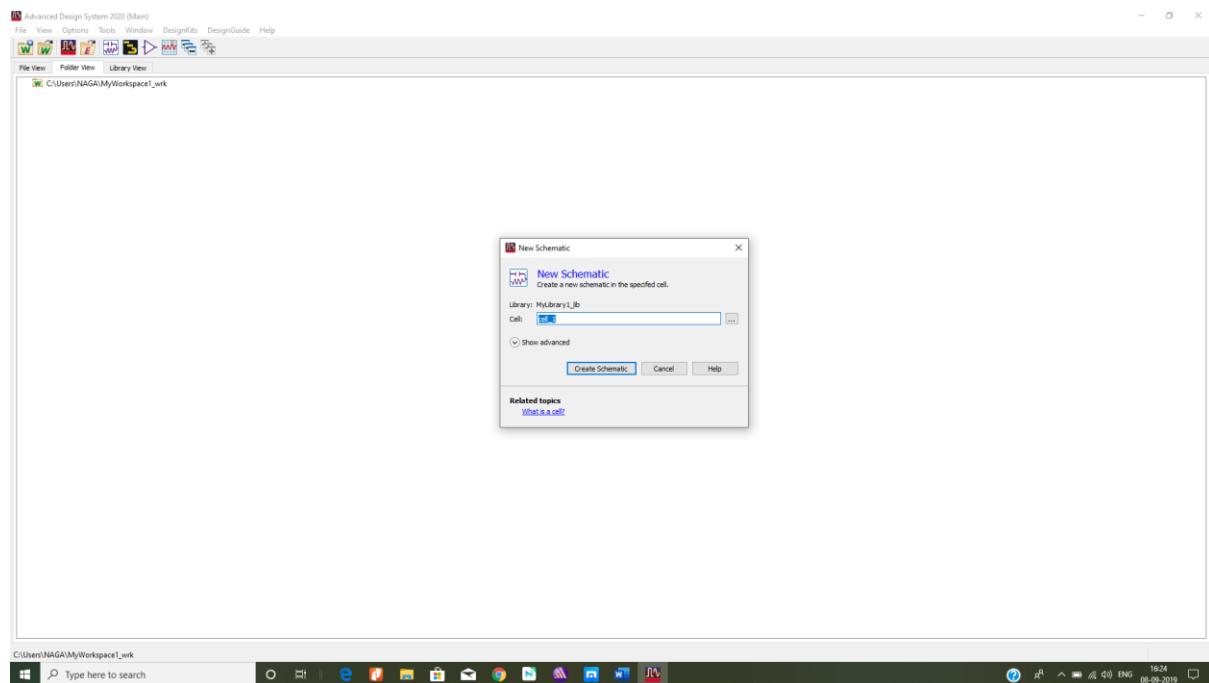
Give the name of the work space in the name box and locate where to create. Click on create button. It will show the interface as shown below



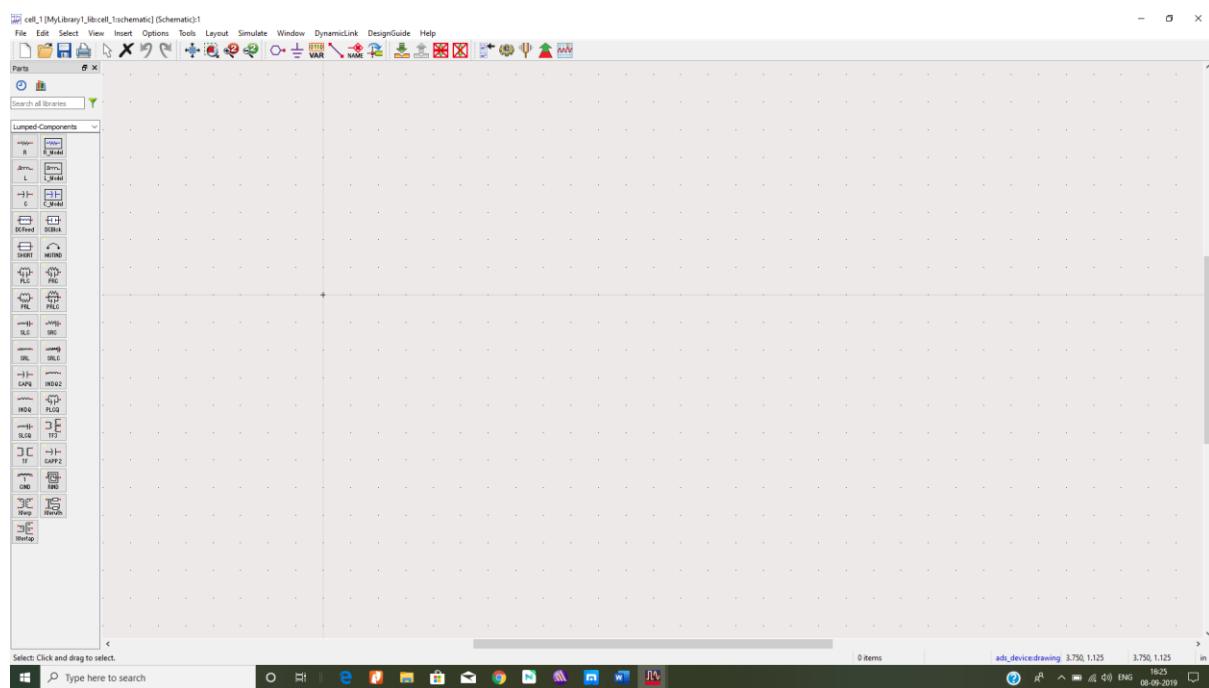
Now create a new schematic by selecting File>New>Schematic



It will show the interface as shown



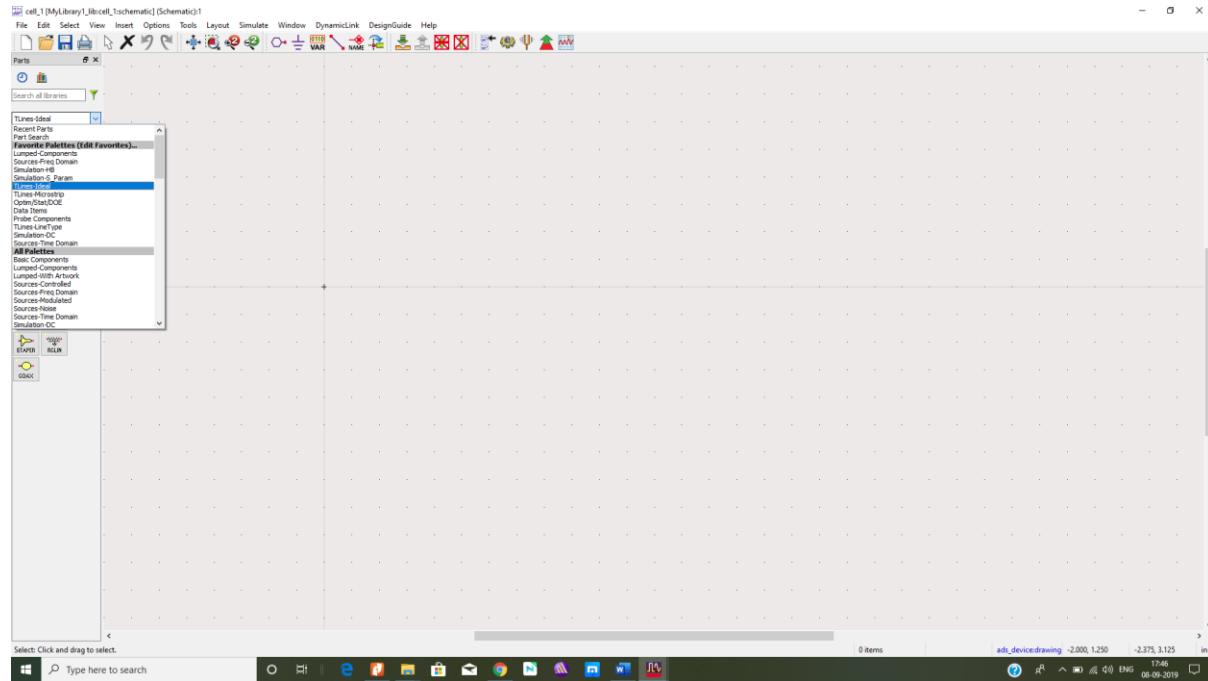
Now enter the cell name and press create schematic. It will give the new interface as below



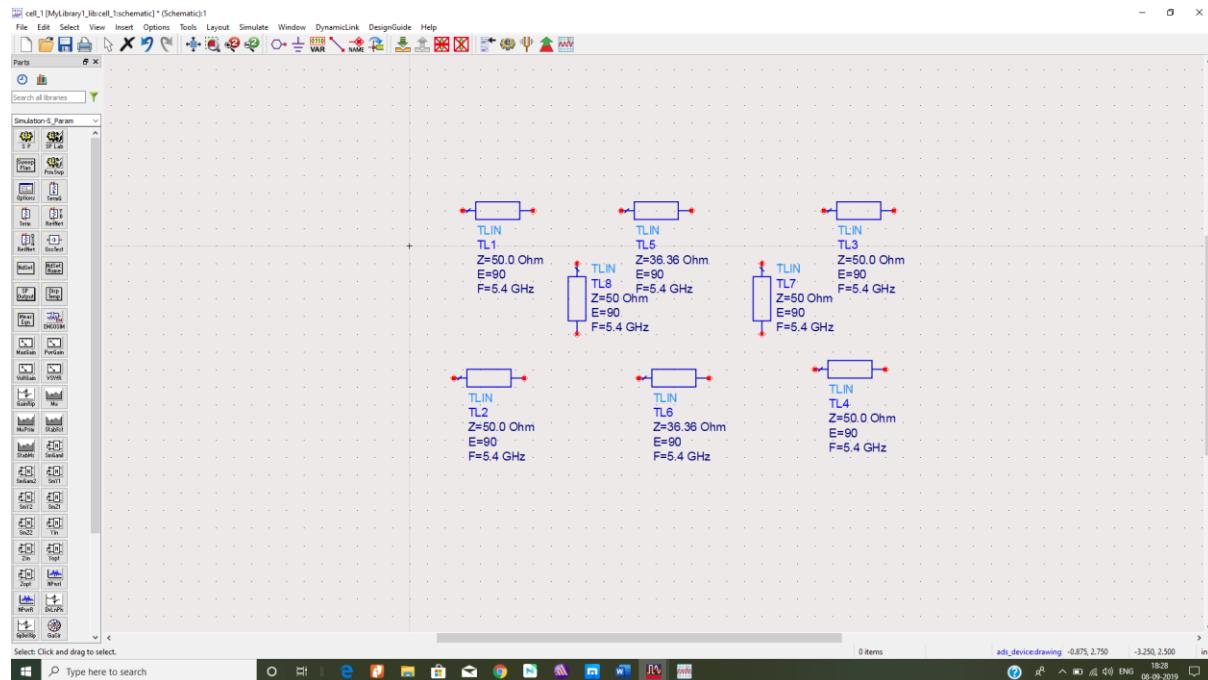
Here we can Design our design

# Design of Directional coupler: (Using Ideal Transmission lines)

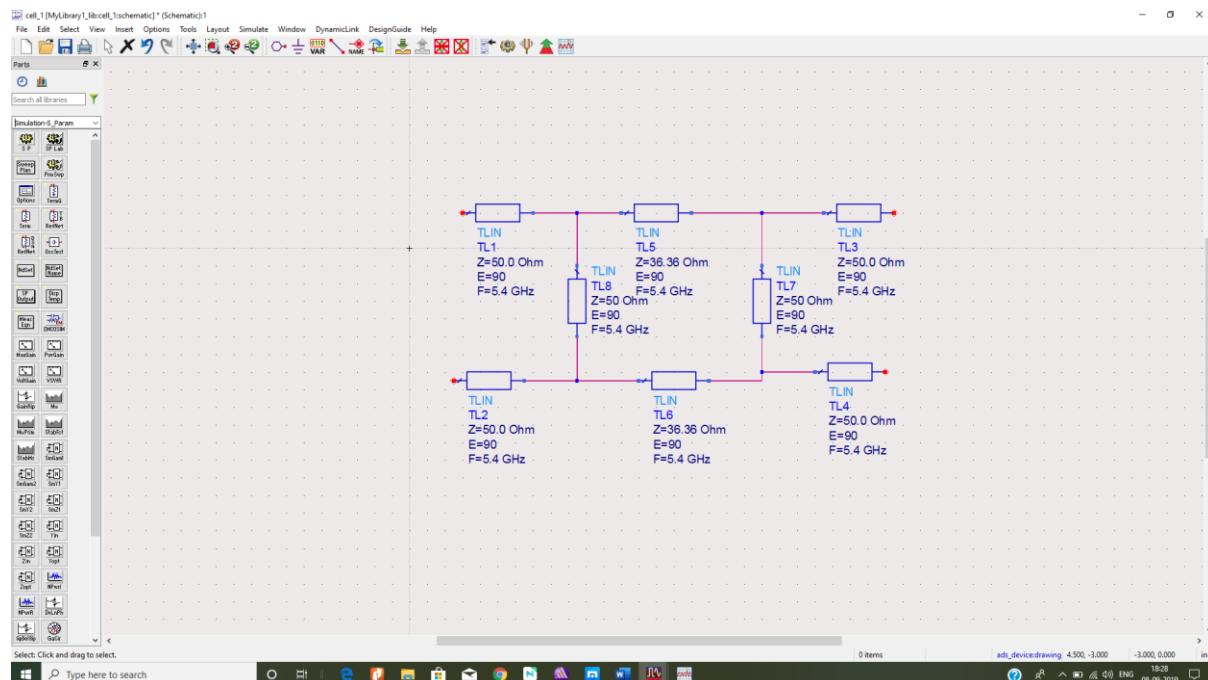
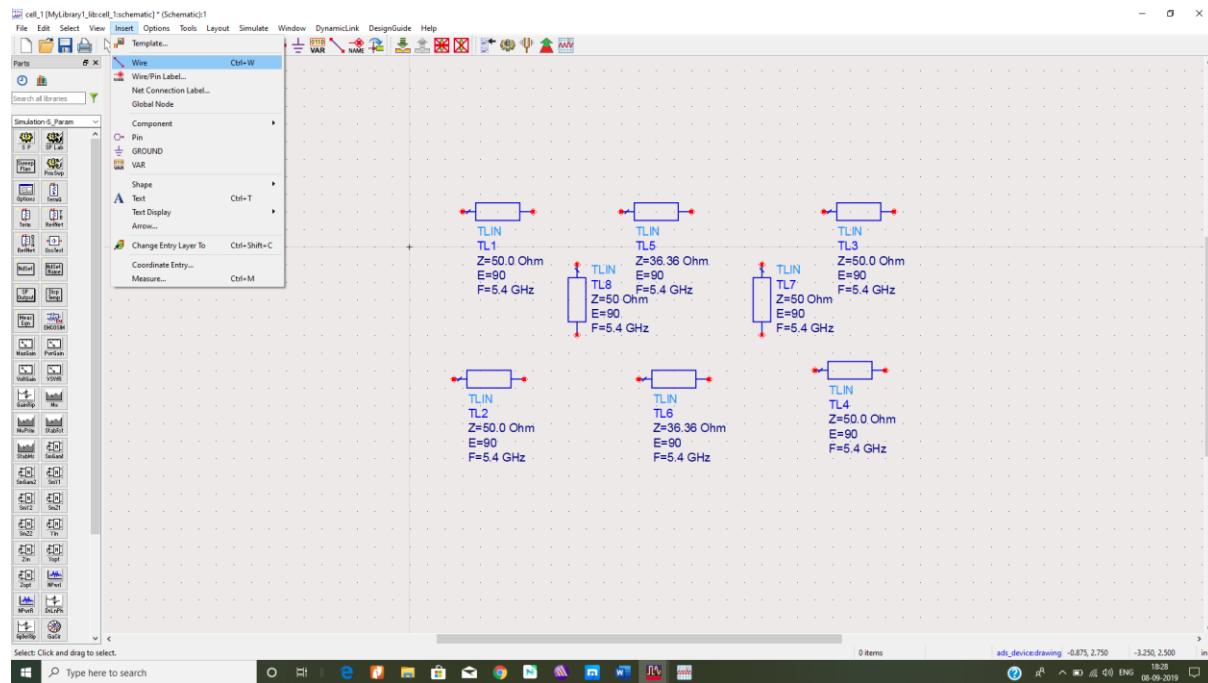
For designing Directional coupler using Ideal Transmission lines, need to select Tline-Ideal form the parts palette.



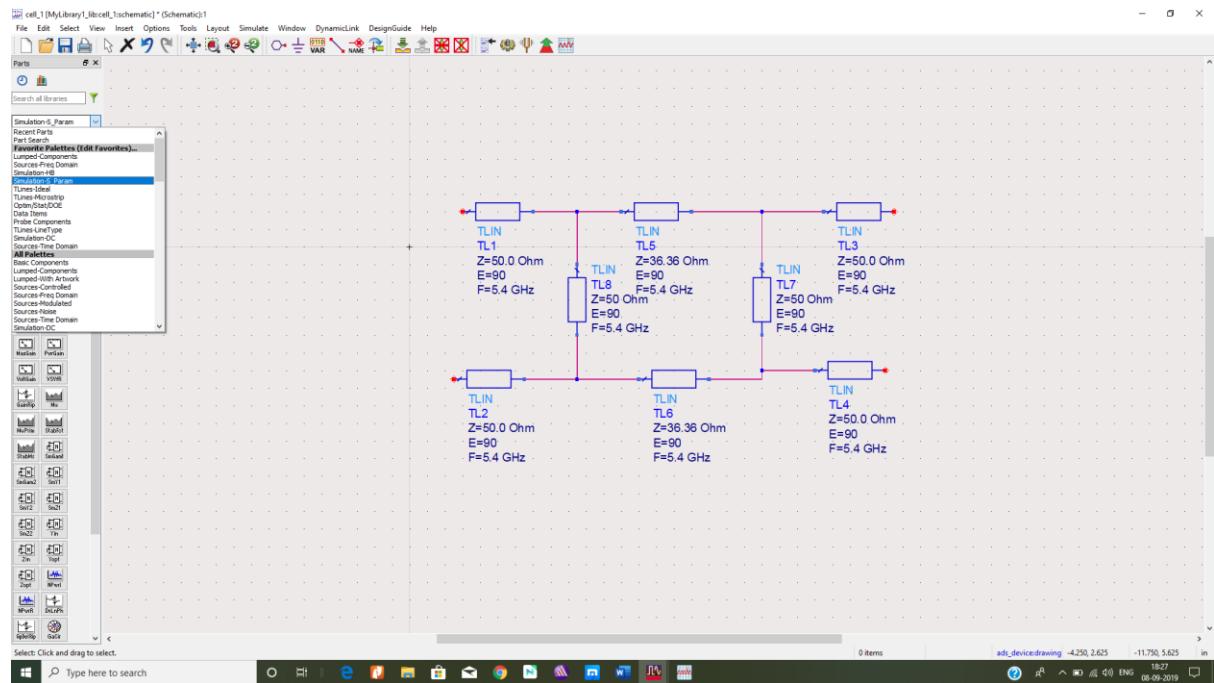
Place all the ideal tx-line with characteristic impedances as required for the directional coupler shown below



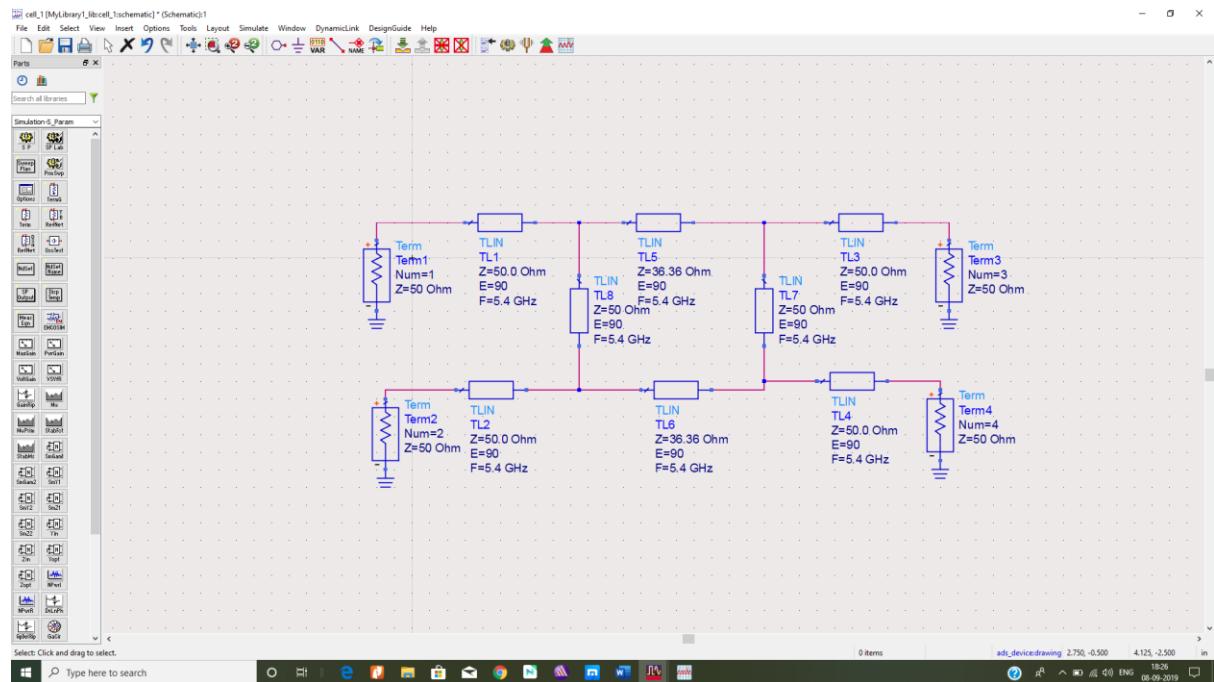
Select insert> wire to connect all the elements.



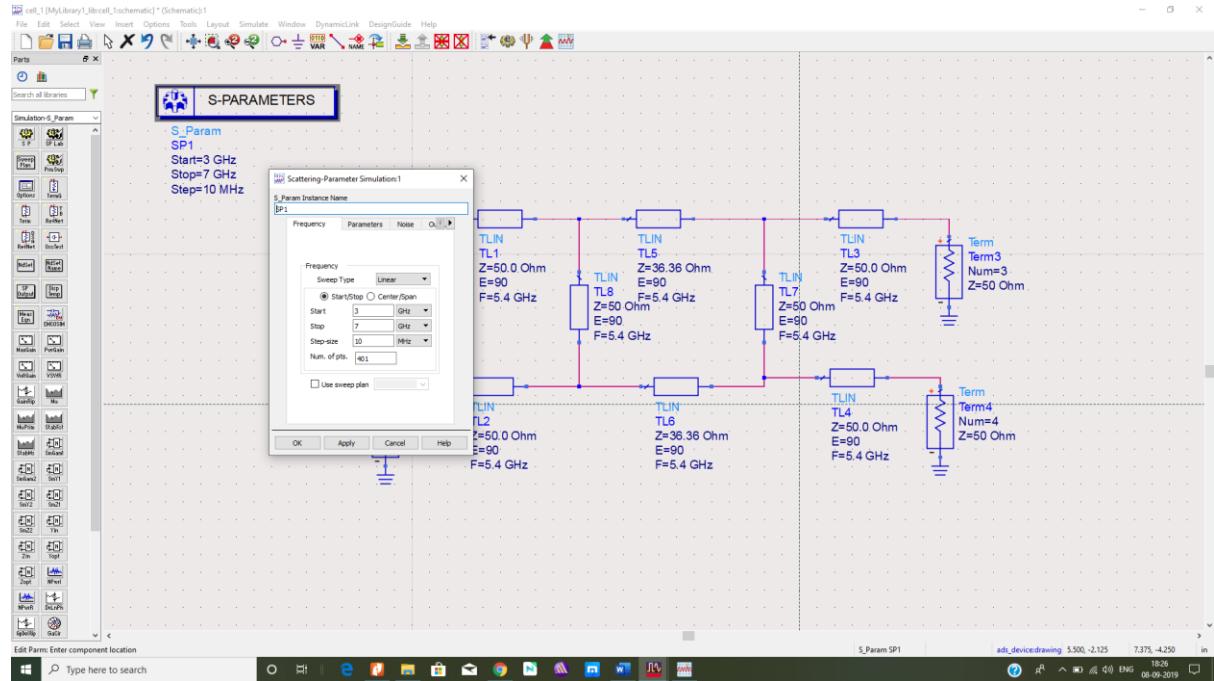
Now for analysis purpose we need to select Simulation-s\_parameters from parts list.



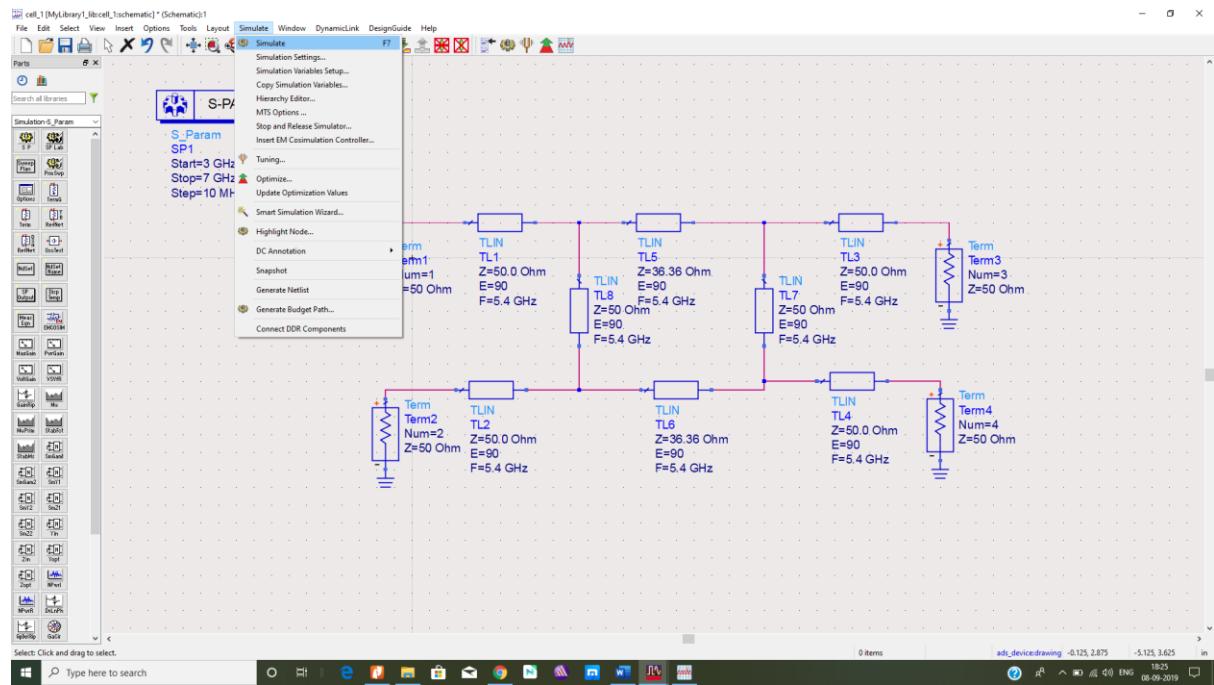
Select the Term element from the palette. And place them as a port termination for the coupler. Connect the negative terminal of Term element to ground.



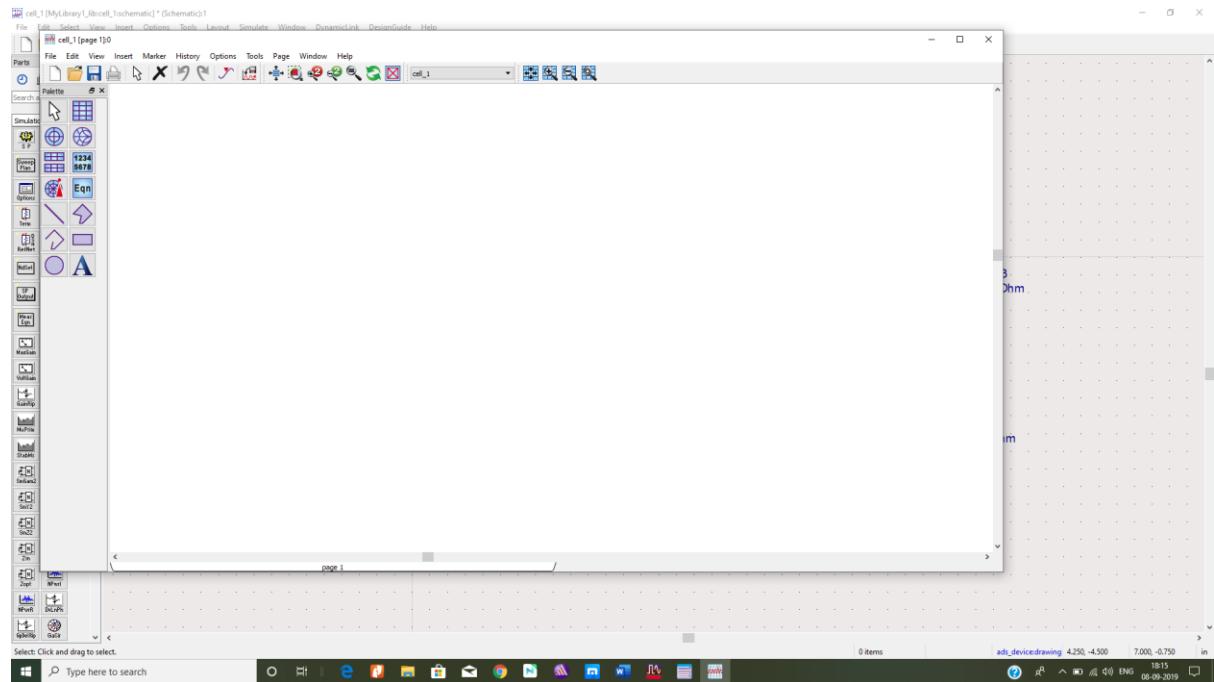
select the SP element from the palette and place it in the schematic. Change the sweep frequency values as required for the simulation.



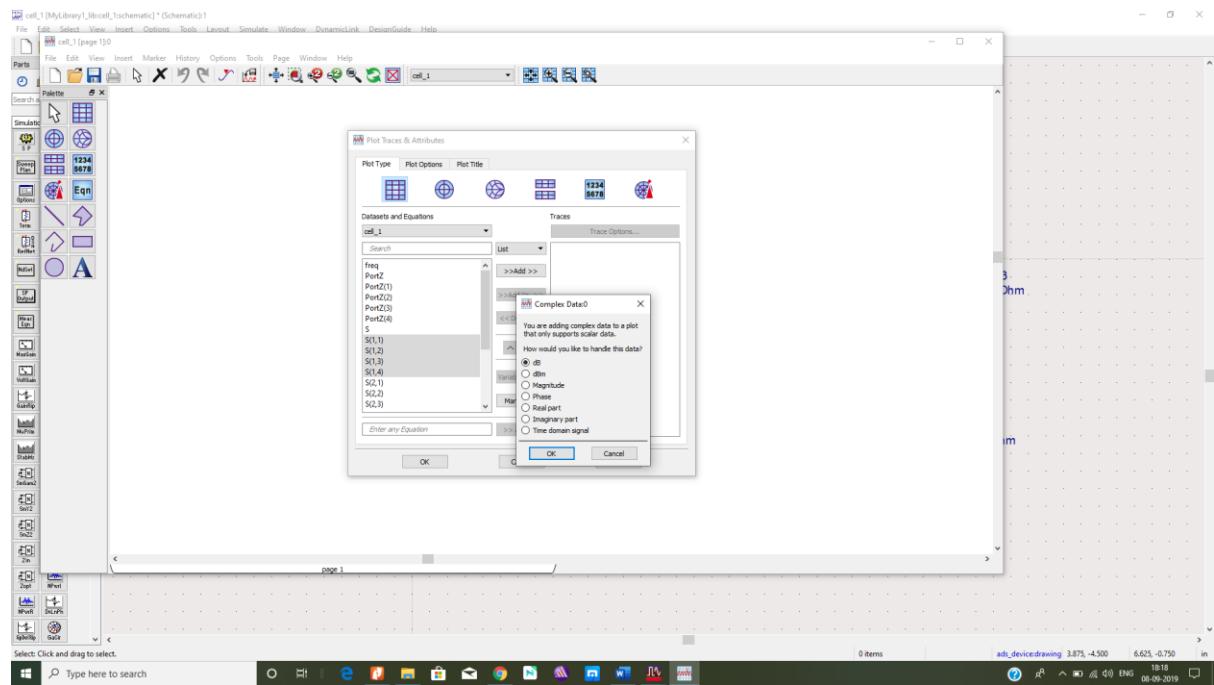
Now go to simulation > simulation or press F7 for simulation of the circuit.



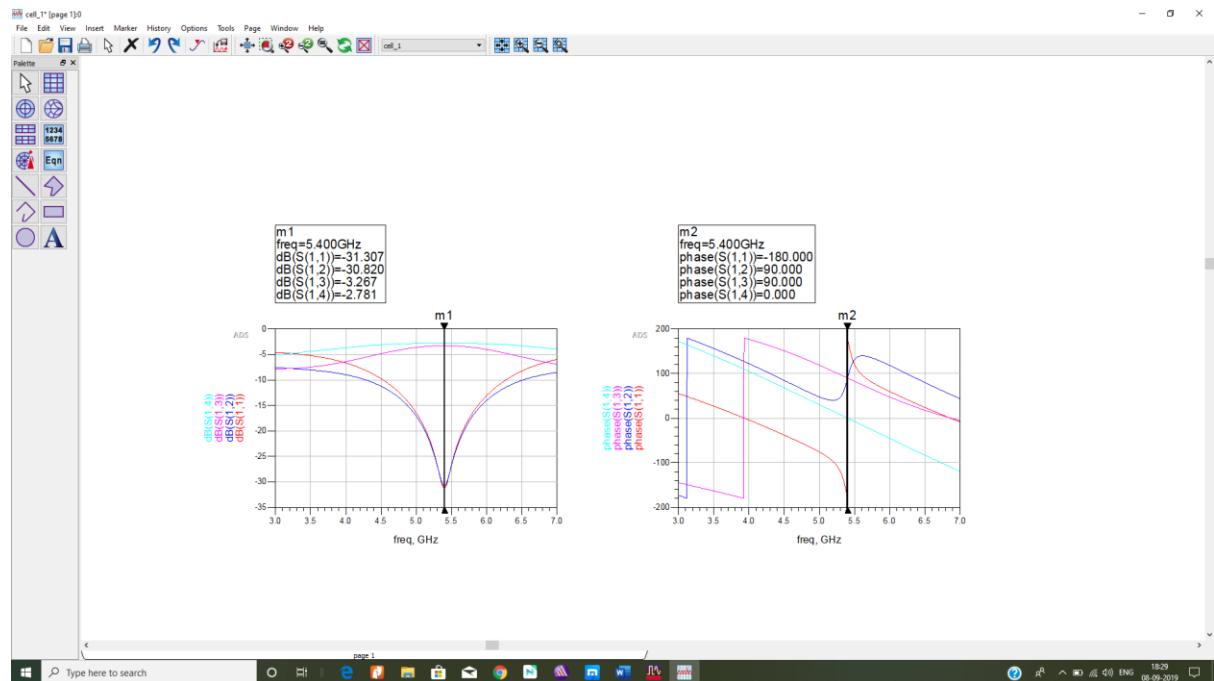
After simulation it will shows the following dialog box



Here we can draw the plots of s-parameters by selecting the rectangular plot and select s-parameters ( $s_{11}$ ,  $s_{12}$ ,  $s_{13}$ ,  $s_{14}$ ) then click add and select in terms of dB then press ok

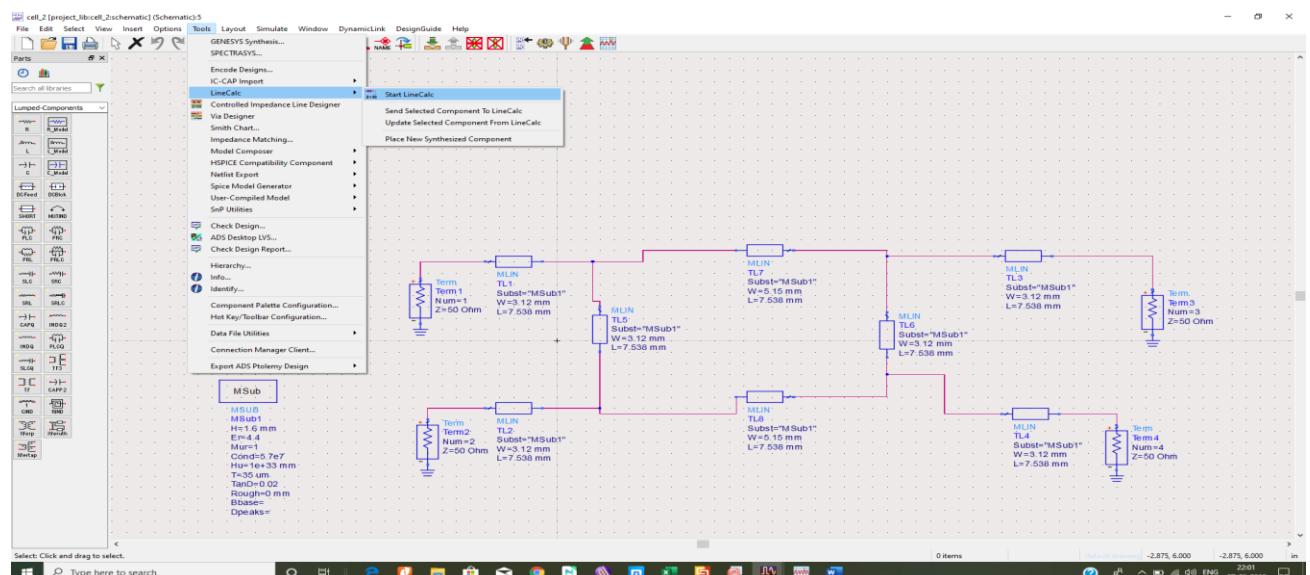


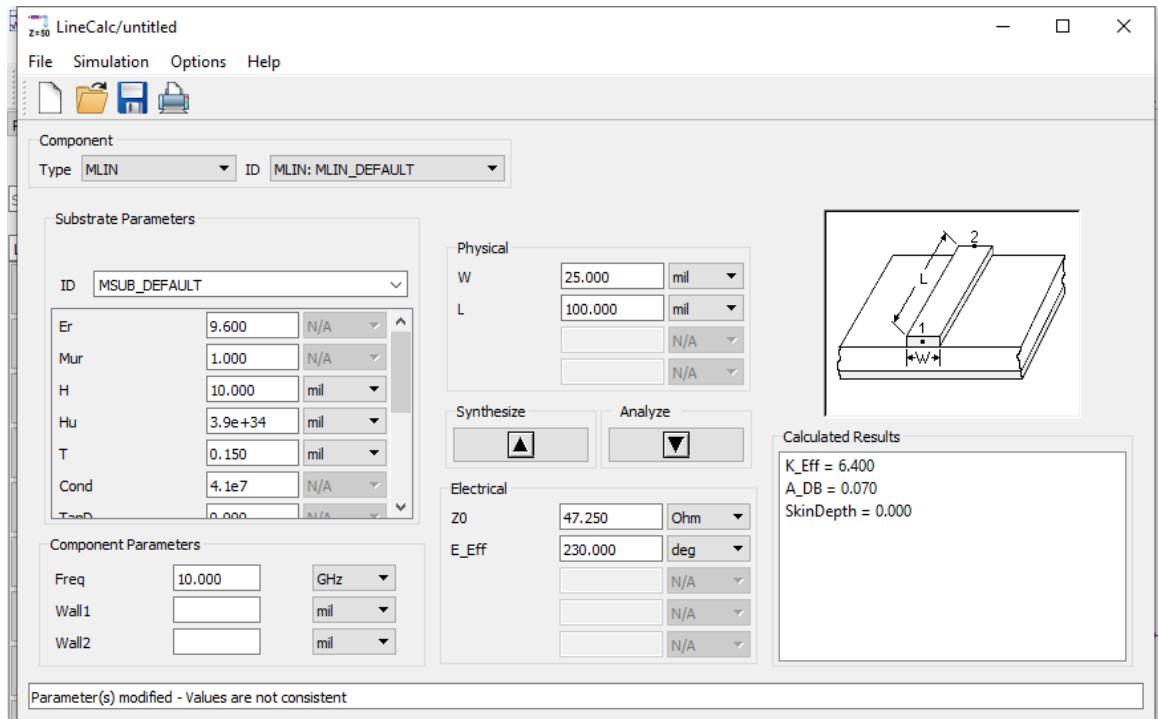
The results for the circuit are



## Microstrip Line Base Design:

For designing microstrip transmission line-based hybrid coupler, corresponding width and length of quarter-wavelength transmission line of impedance  $Z_0$  need to be calculate. For this calculation we can use a tool “linecal” from tools menu.

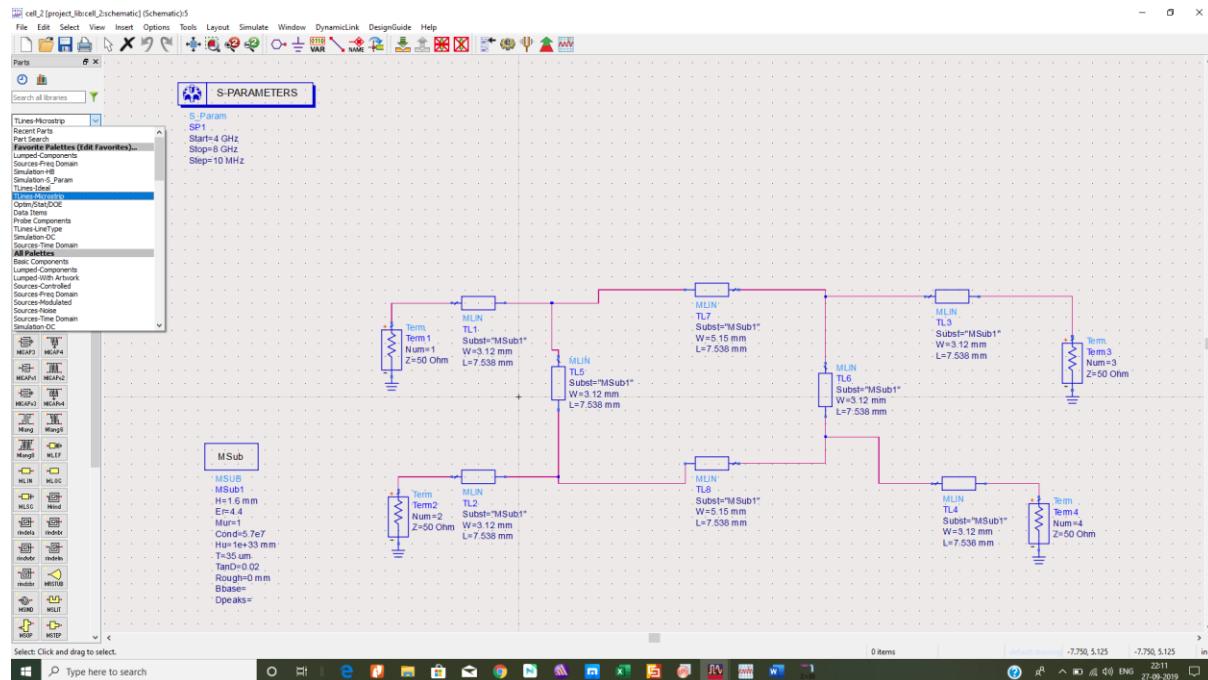




This window will open, when we open tools>linecal>start linecal. If we provide the substrate parameters and required impedance and effective electrical length, and press synthesize button, it will provide width and length of the microstrip line.

With the help of this tool, calculate all the widths and lengths required for the design.

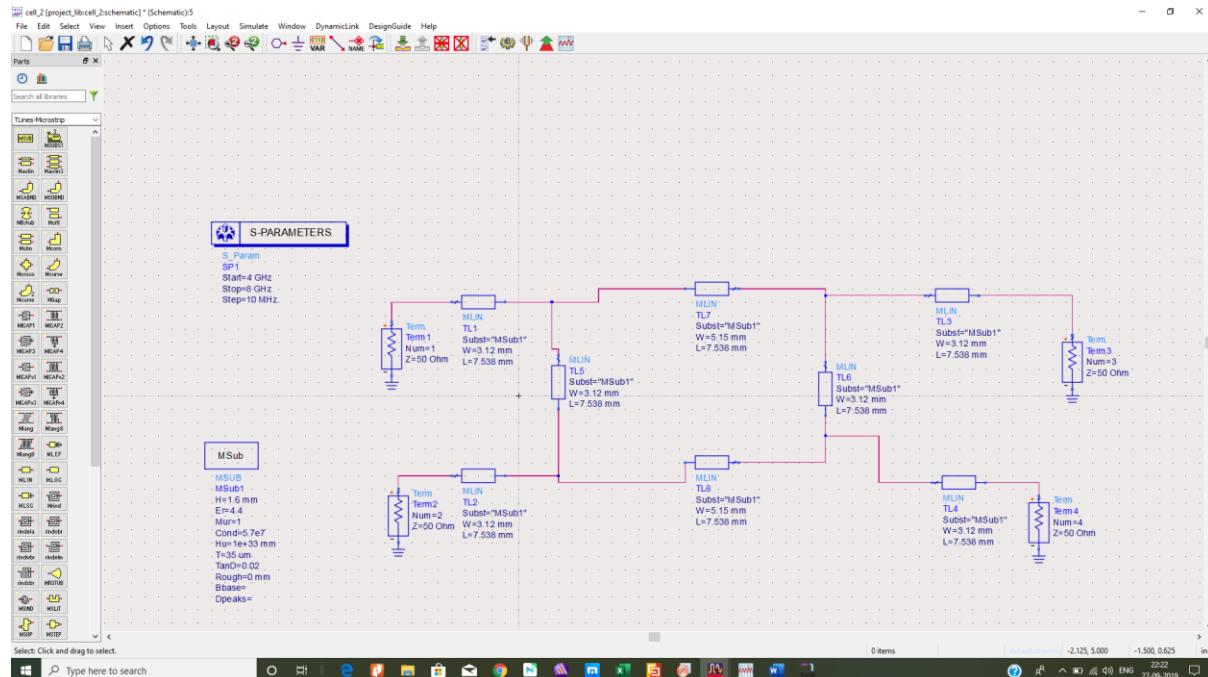
Now open the schematic window, and from the left palette, select the Tline-Microstrip.



From the left palette, select the MLIN and place it in the schematic and change the width and length as required and use all the elements properly to form hybrid coupler. Connect them

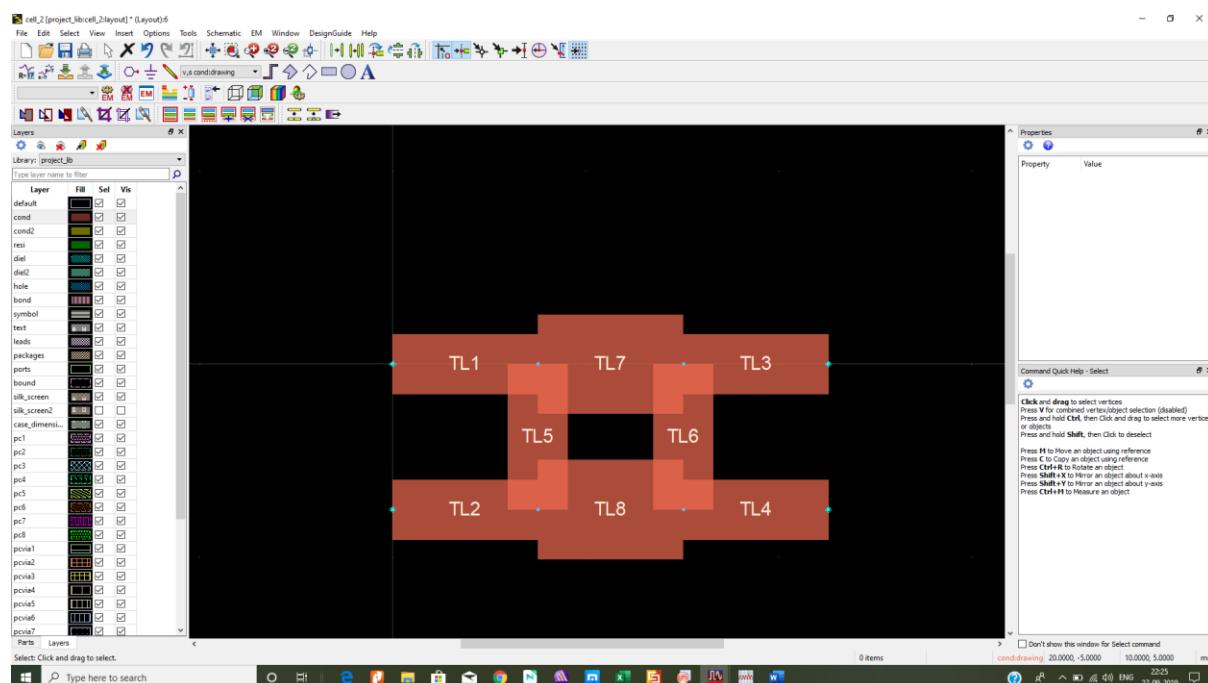
using wire. Place the Msub element in schematic from the palette and provide the substrate parameters.

For analysis purpose, select simulation s-parameter from the left palette, and place the TERM element at the input and output terminals and terminate it with the GND element. Place the S-parameter analysis in the schematic as shown below.

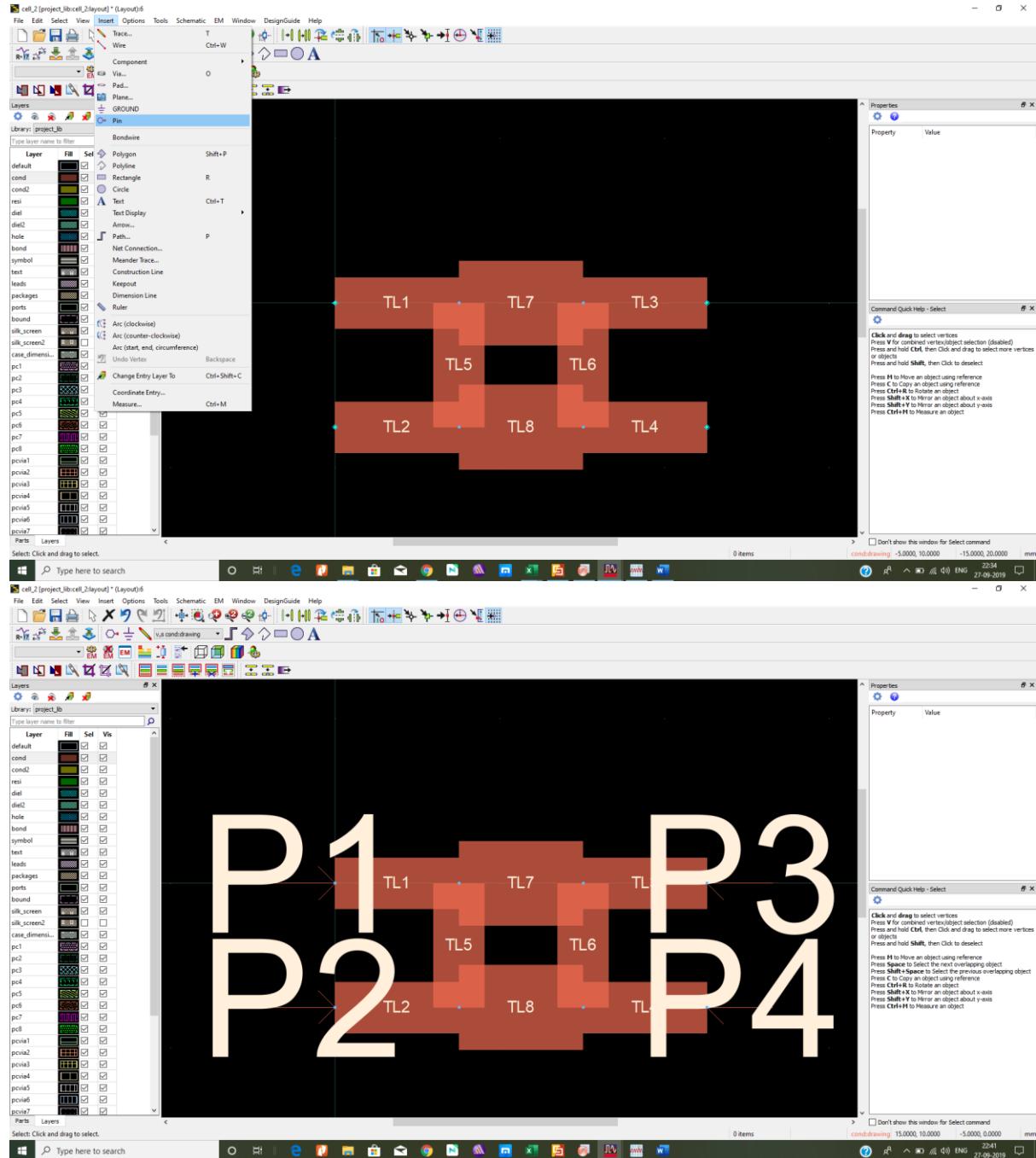


Now simulate the design using simulate>simulate.

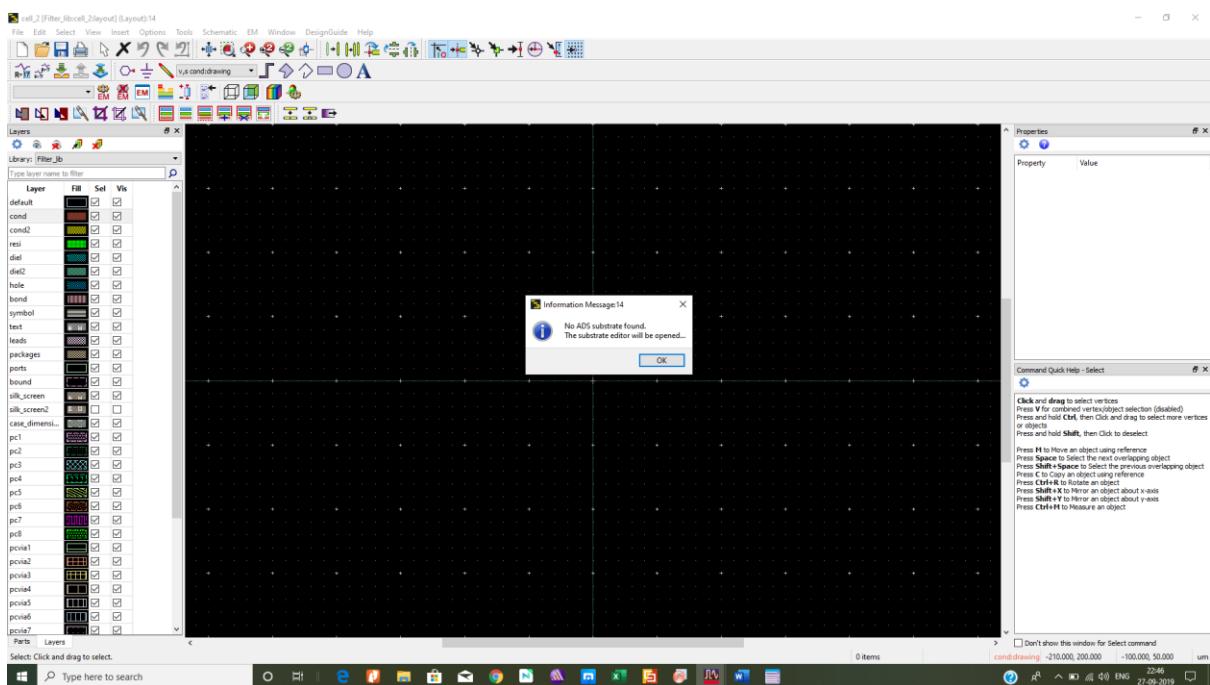
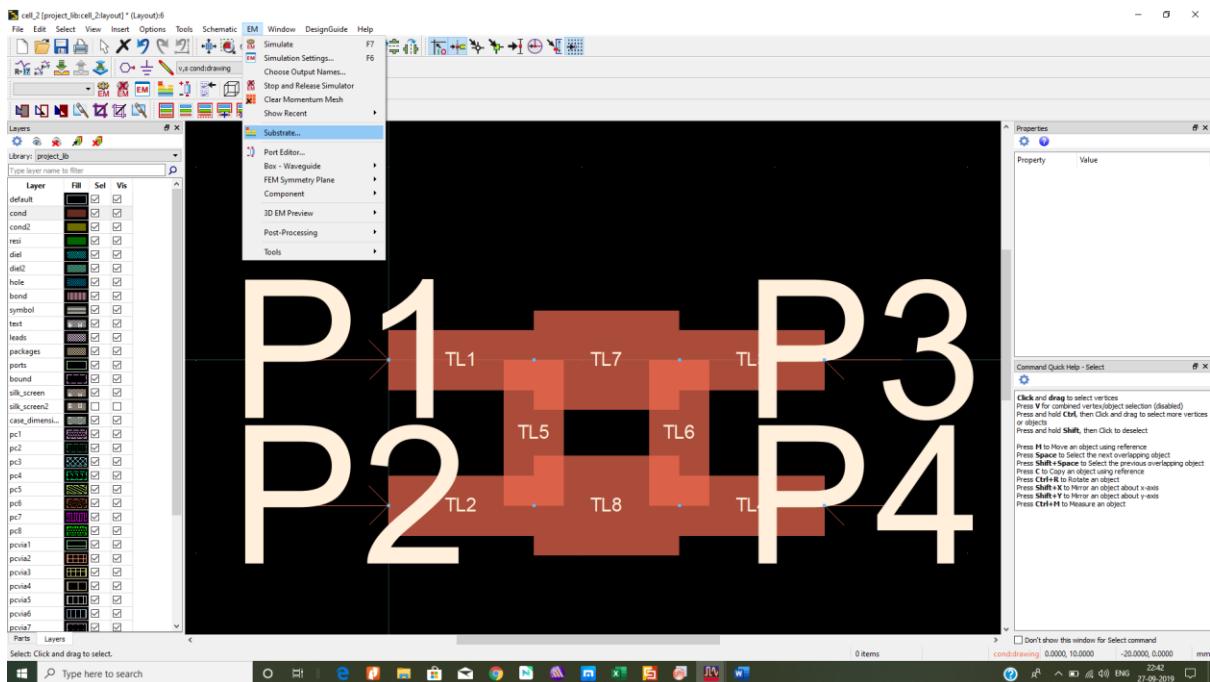
For generating the layout go to layout> Generate/Update layout. It will generate the layout of the corresponding schematic.



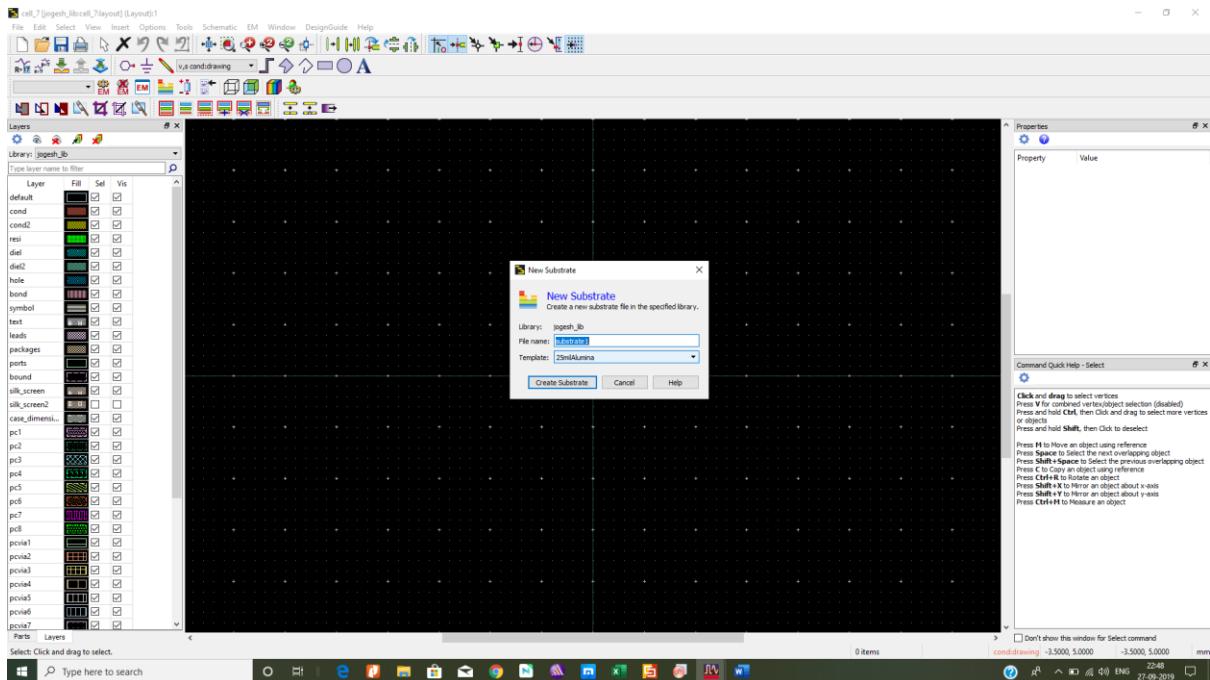
For simulating this layout, at the input and output, provide the pin by selecting **tools>pin**



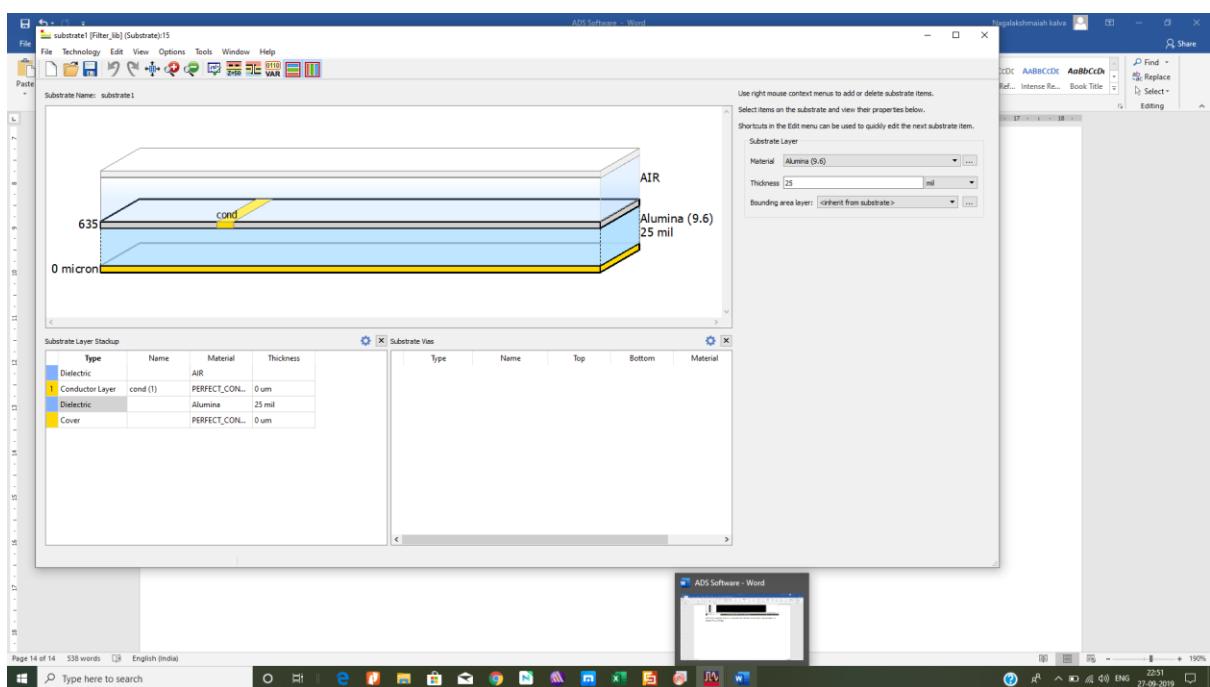
Now we need to define the substrate for the simulation of layout. Go to **EM> Substrate** and define the substrate parameters as shown below.



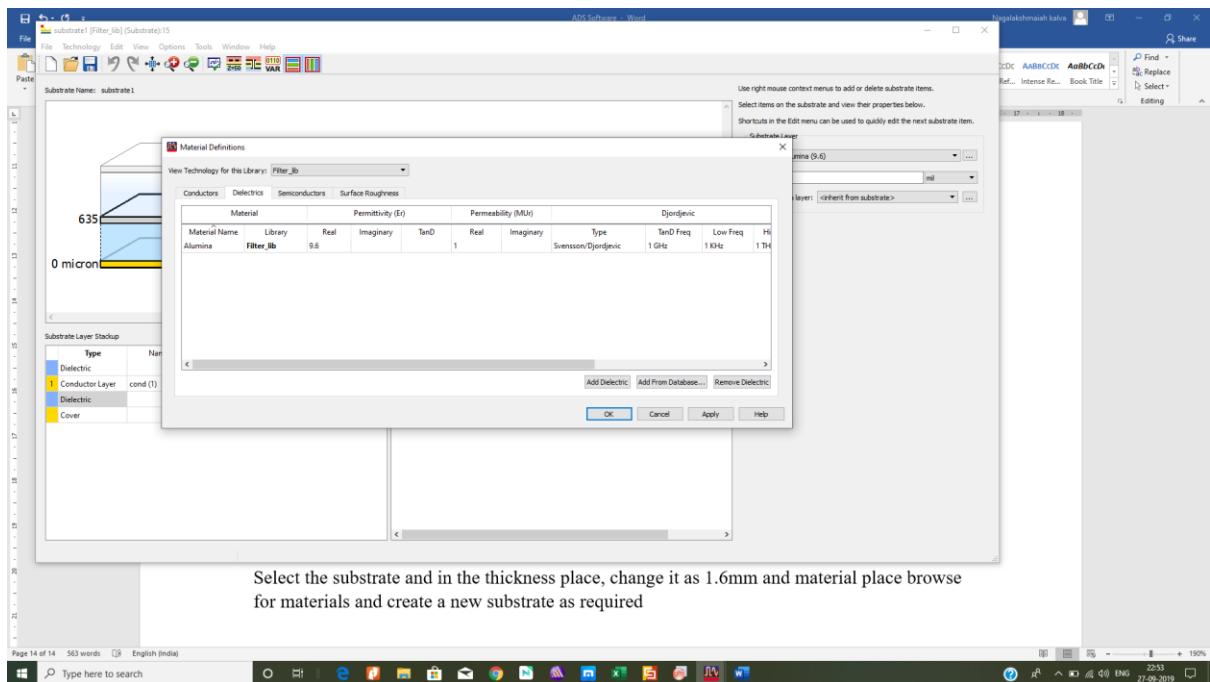
Then press OK.



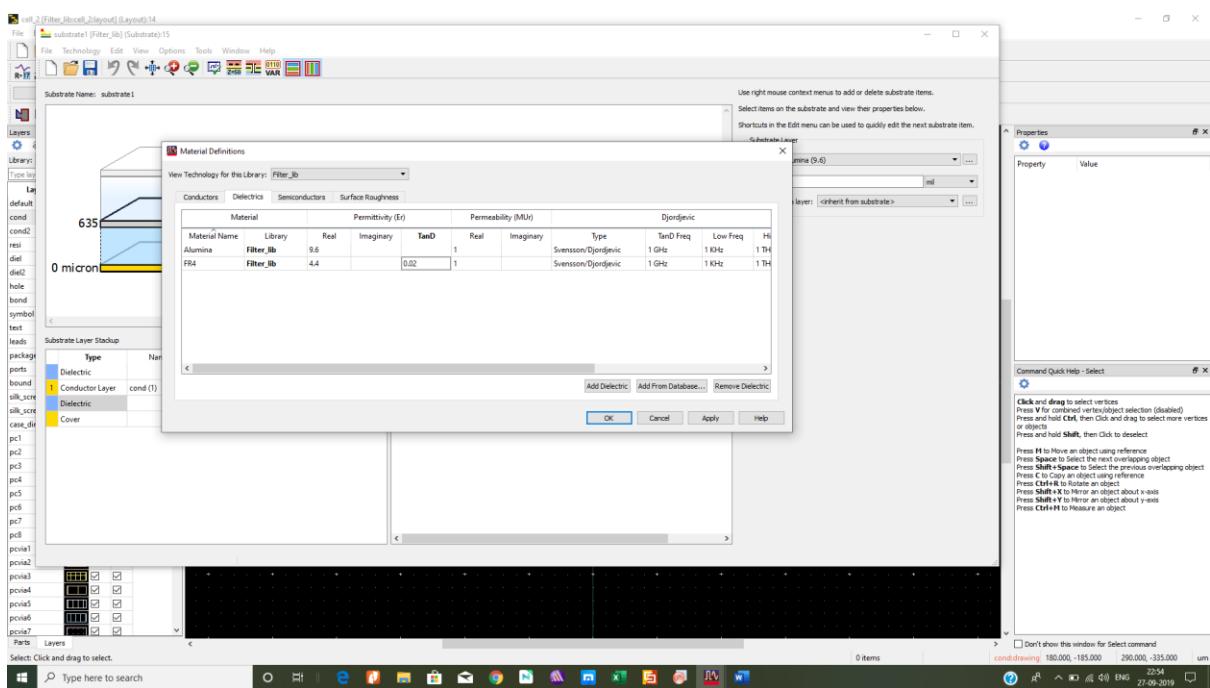
Select 25mil alumina and press create substrate and then we can replace the parameters as required for our design.

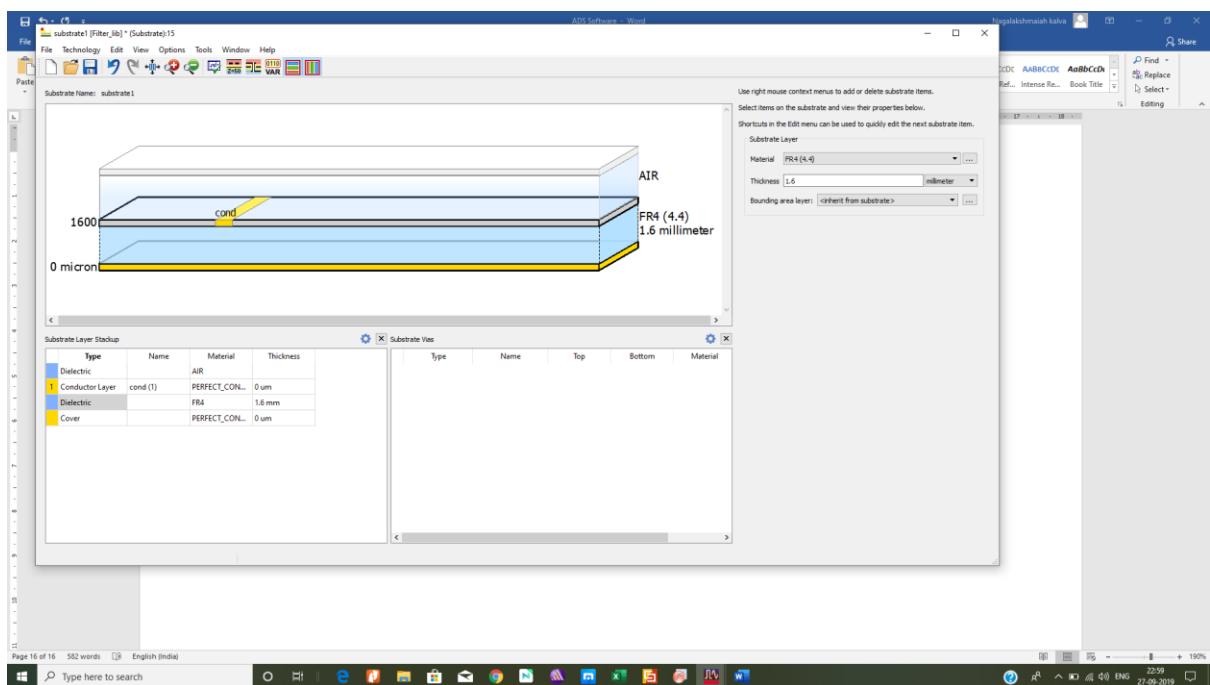
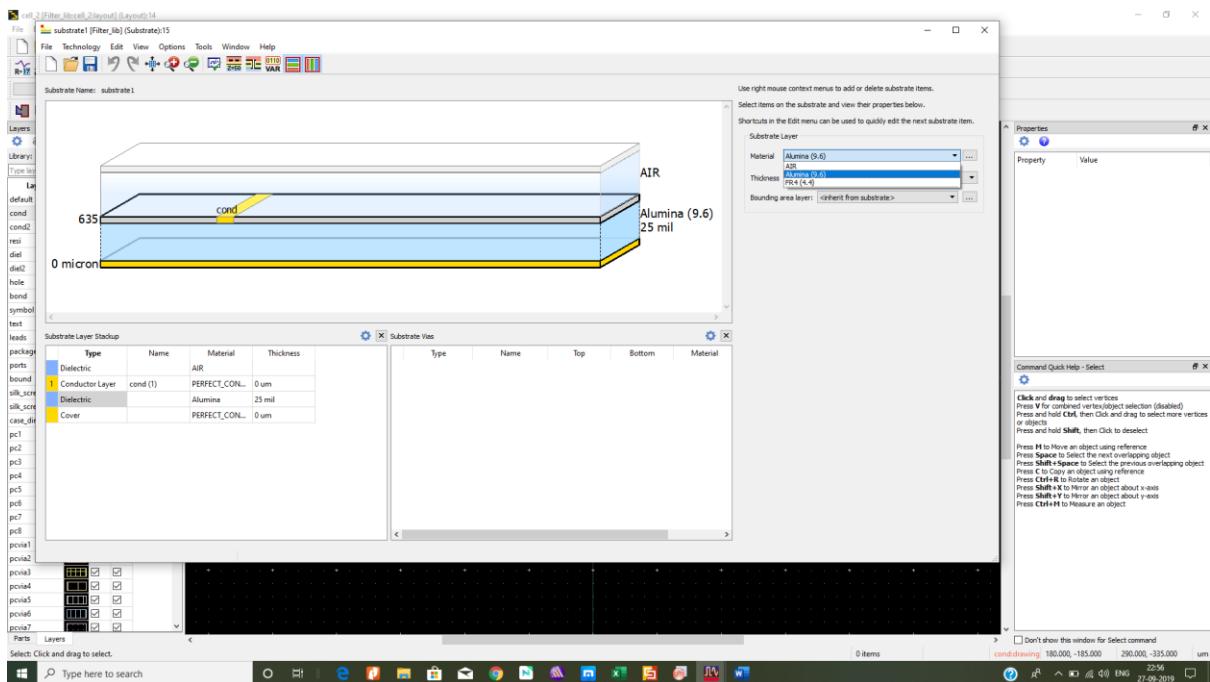


Select the substrate and in the thickness place, change it as 1.6mm and material place browse for materials and create a new substrate as required

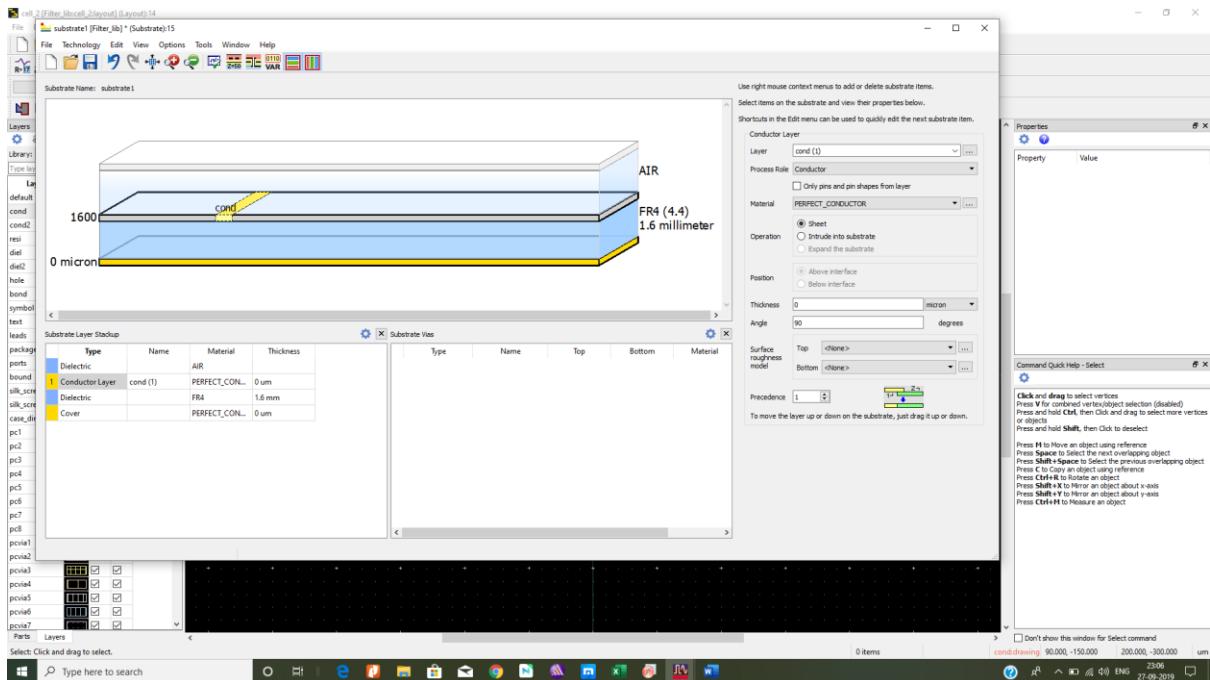


Enter the parameters as required and press OK

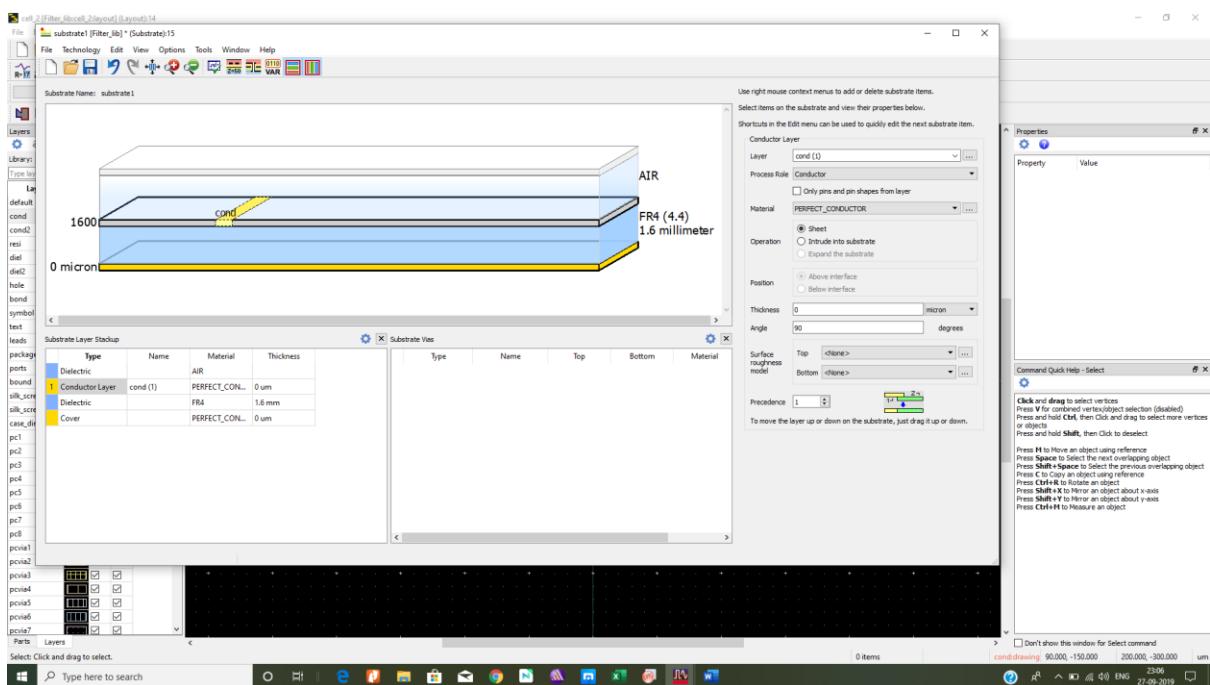


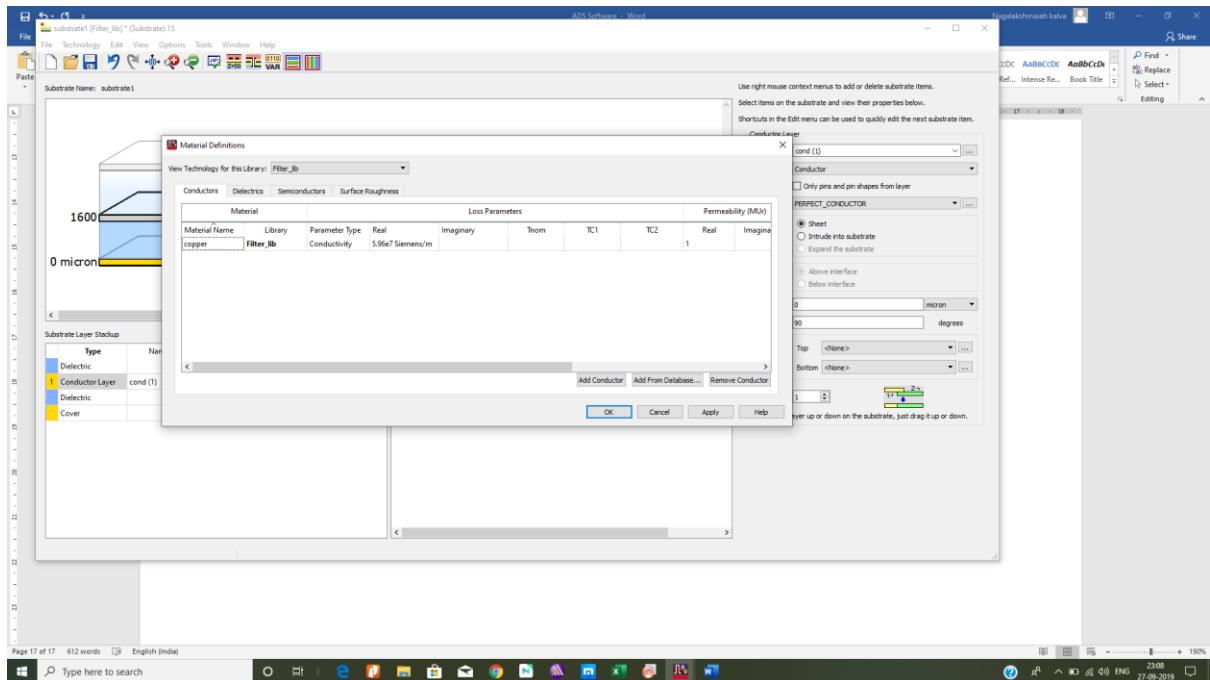


Now select the top and bottom conductor and change the properties as below.

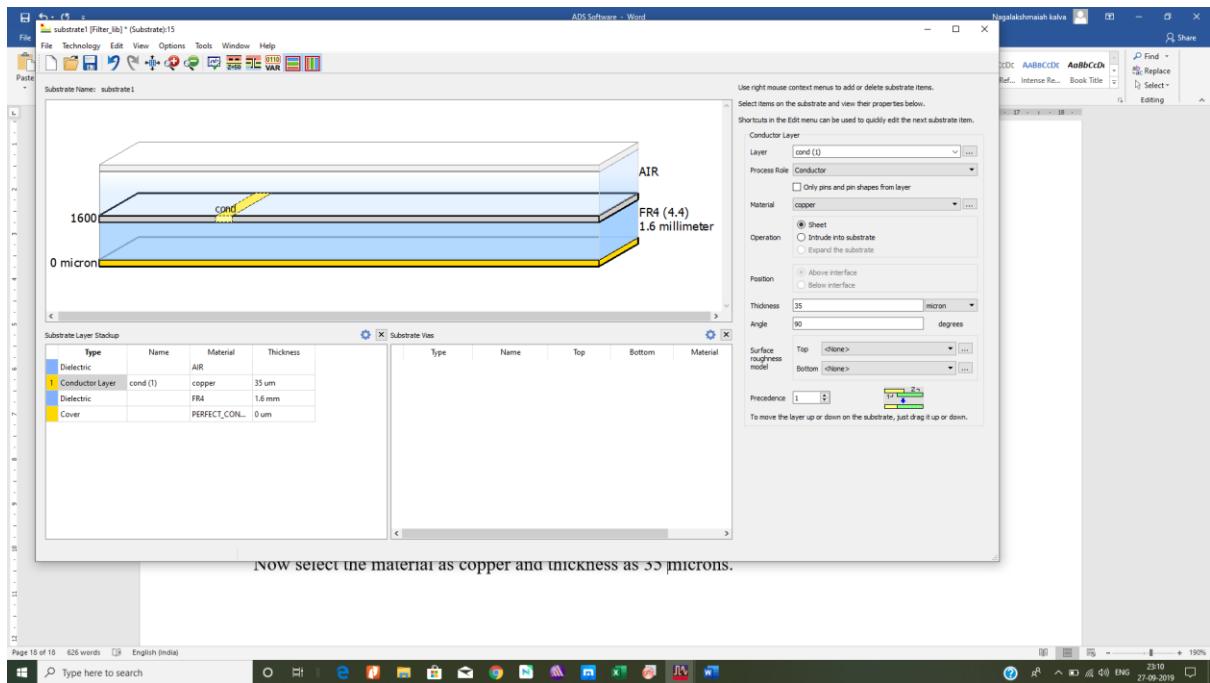


Now select the material, add conductor copper and give the conductivity of the copper material as  $5.96 \times 10^7$  and press OK.



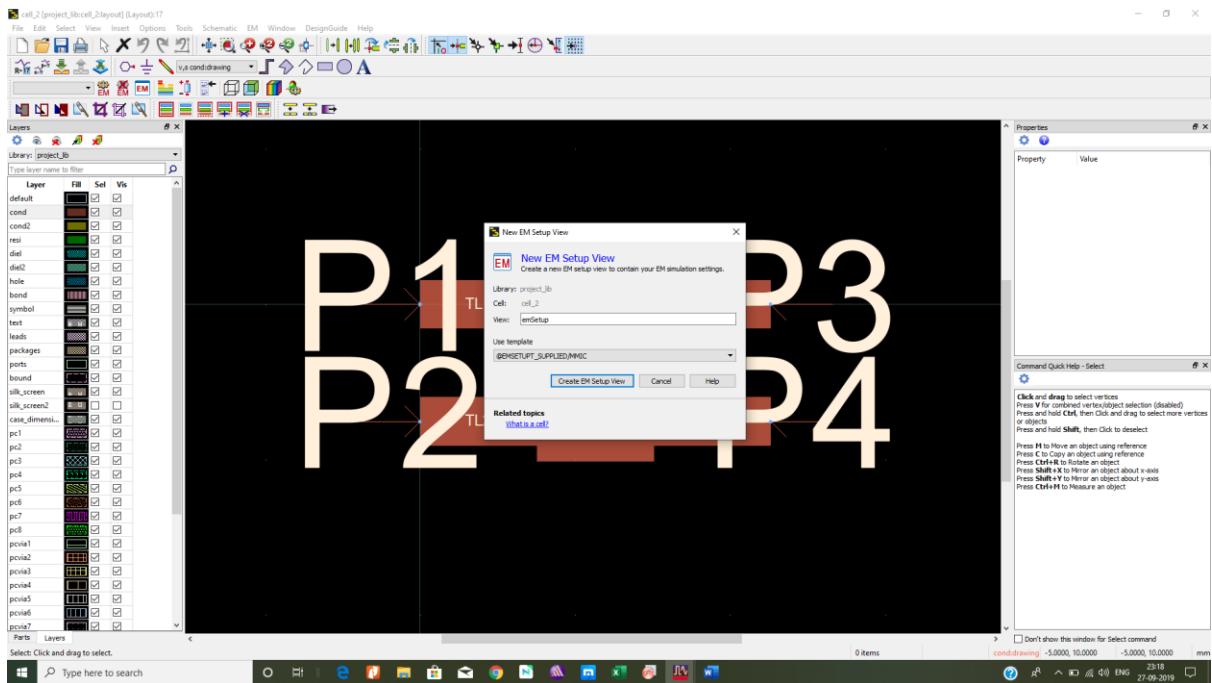
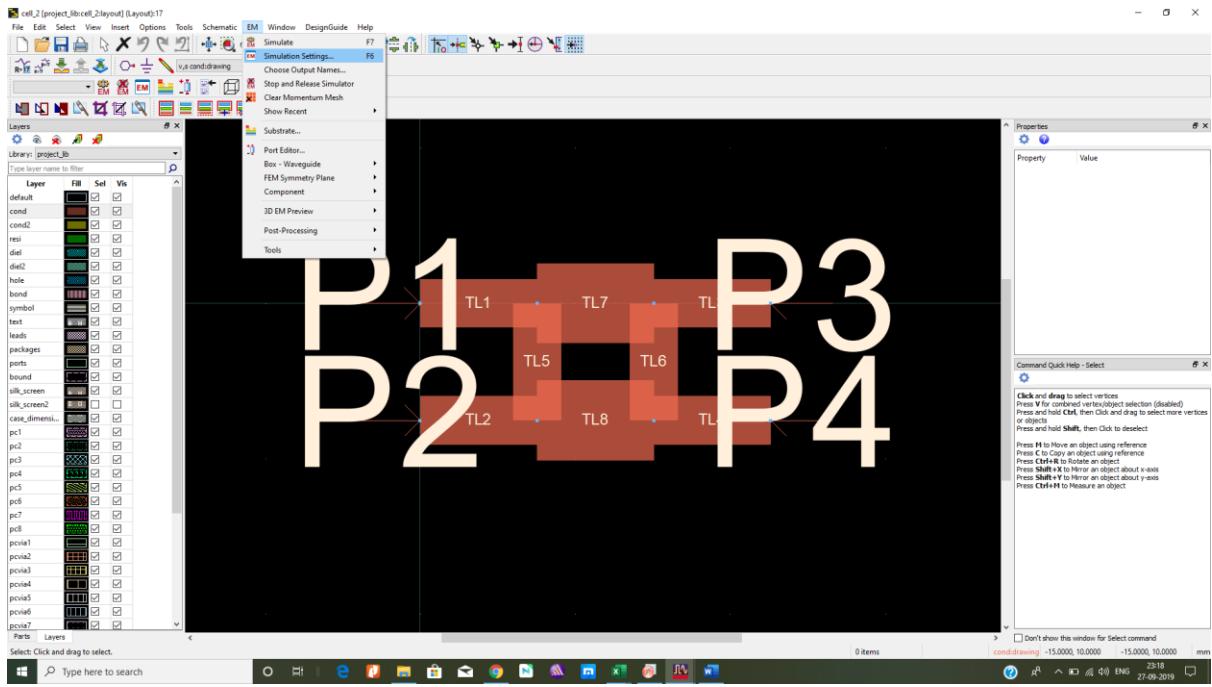


Now select the material as copper and thickness as 35 microns.

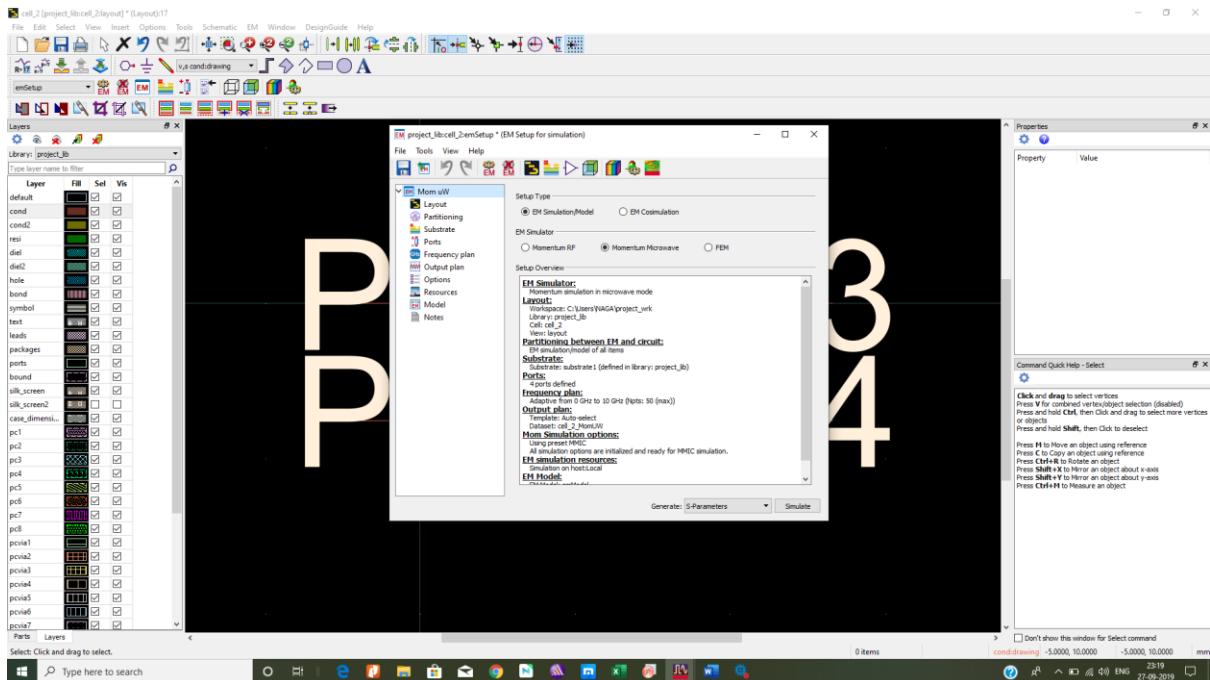


Then do the same for bottom conductor and then save the substrate.

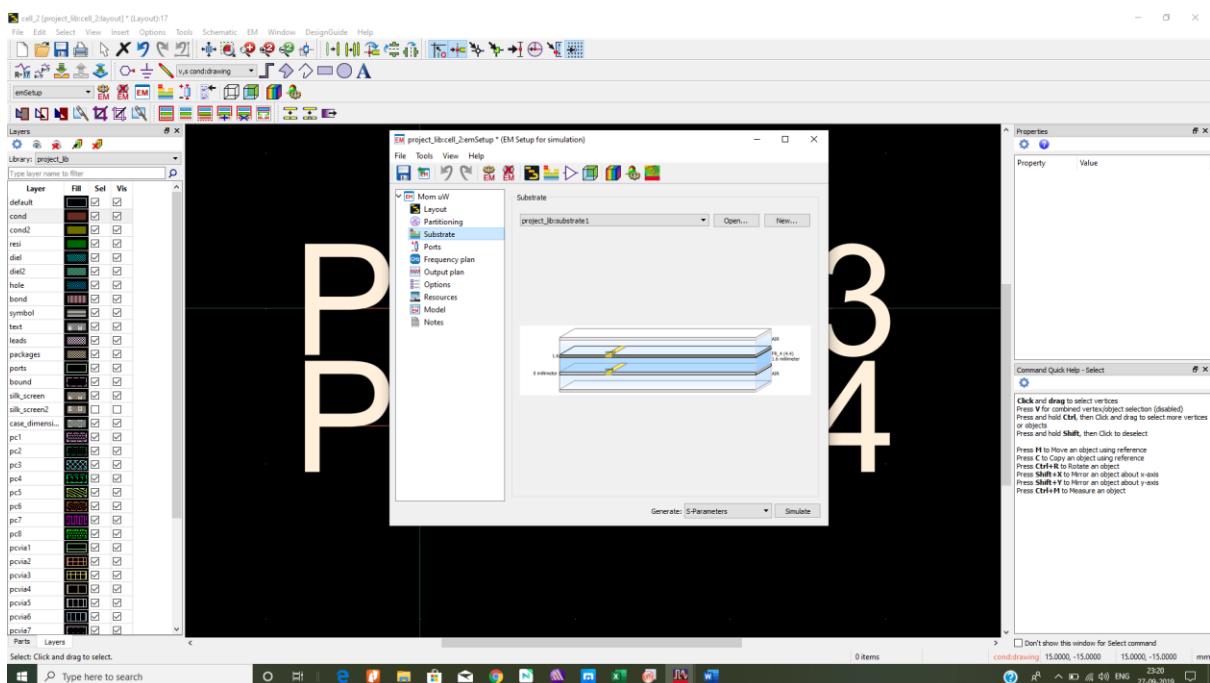
Now for simulating the layout goto EM>simulation settings



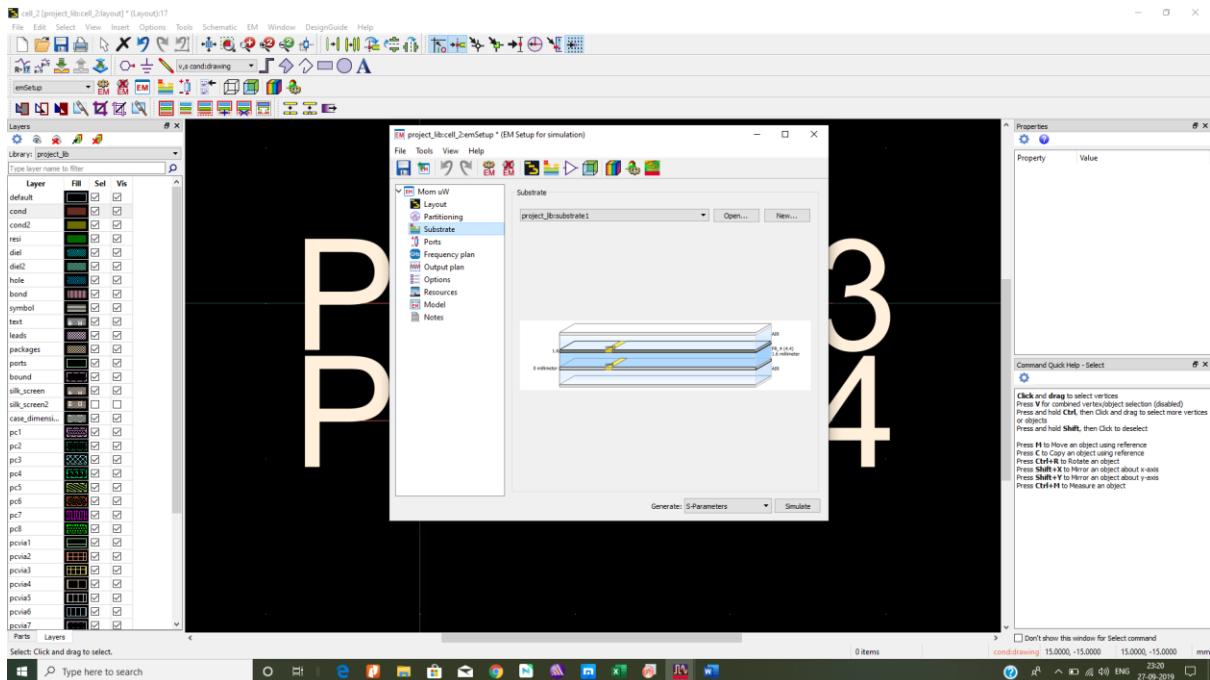
Select create EM setup View



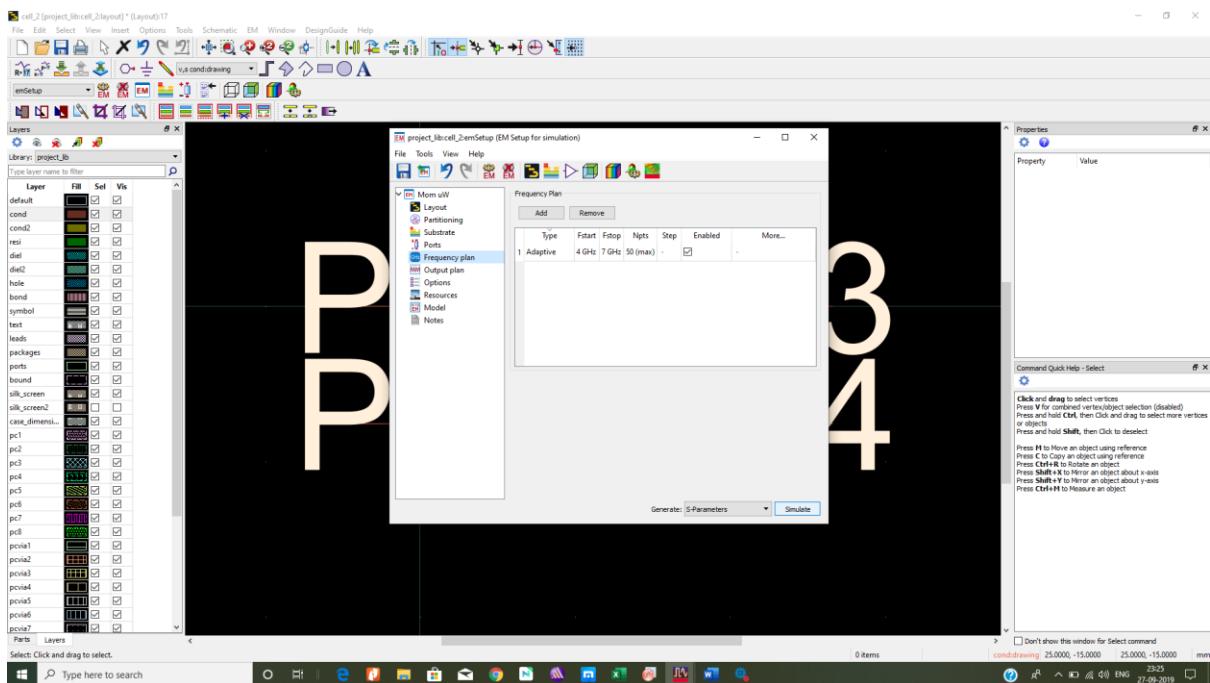
Select the substrate which is created before.



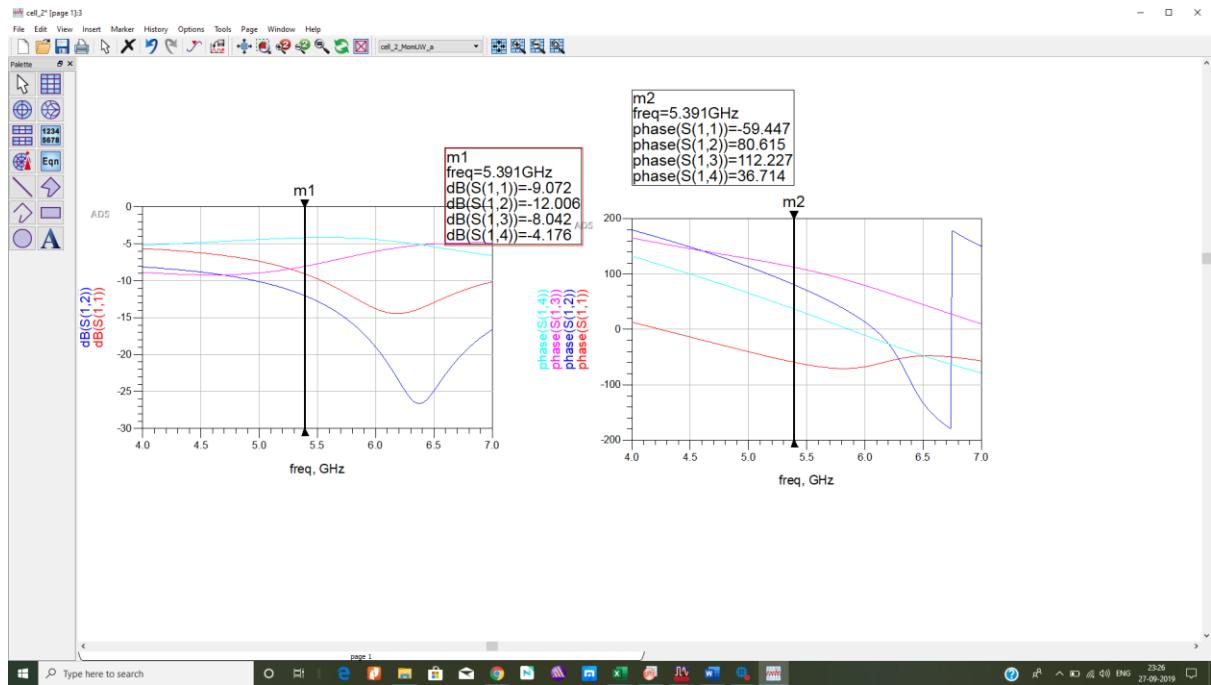
Select the frequency plan and give the frequency range as required.



Select S-parameter and Press the simulate which is present below the simulation settings.



After simulation of the layout, it will open the results as below.



Now verify the results and reiterate the procedure and optimize the schematic and layout for the required results.