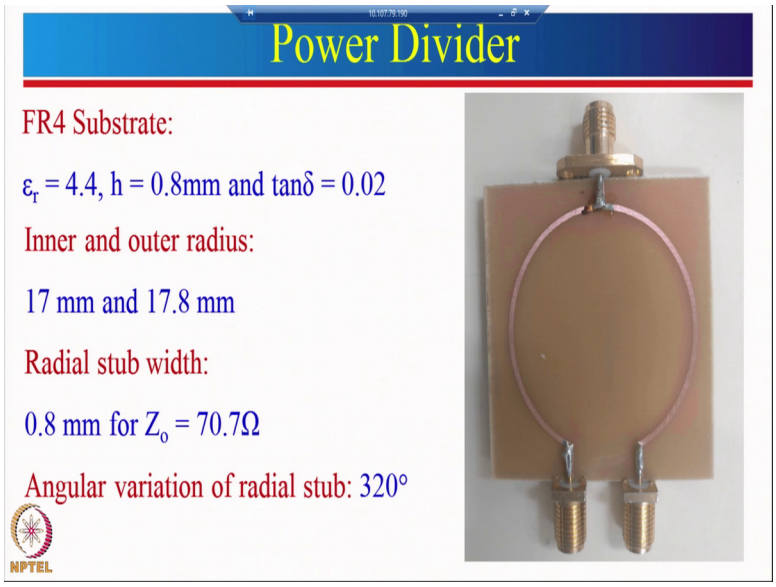
Microwave Theory and Techniques
Prof. Girish Kumar
Electrical Engineering Department
Indian Institute of Technology, Bombay

Module - 12
Lecture - 57
Hybrid Coupler Design

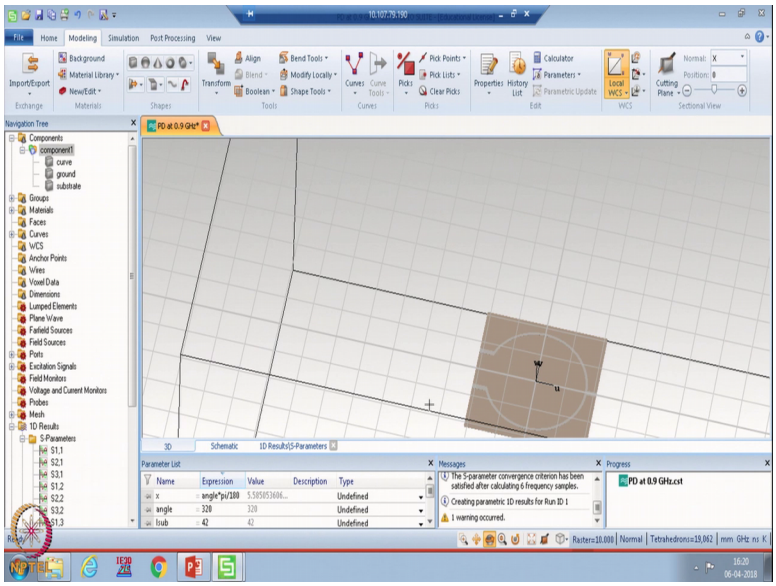
Hello, in the last lecture we tried to design Microstrip Bandpass filter and band reject filter then we saw how to do the parametric analysis, if we want to change a particular parameter, after that we tried to design a 2 way equal power divider.

(Refer Slide Time: 00:42)



So, this is the geometry of 2 way equal power divider, this we tried to design in CST microwave studio on FR4 substrate and then we saw the simulated results. So, let us go to the CST microwave studio.

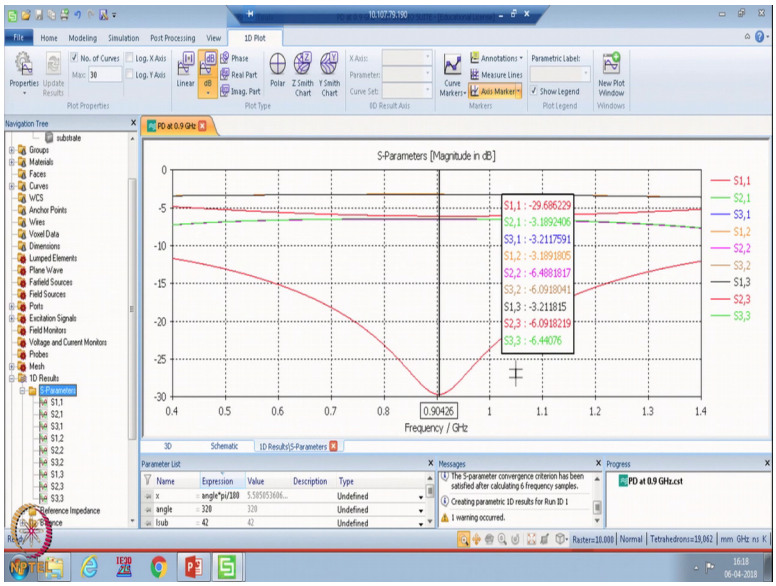
(Refer Slide Time: 00:59)



So, you see this is a geometry that we created in the last lecture of 2 way power divider. We simulated this design to see its parameter, and then we saw when the output is given to port 2, the power at port 3 is very high. So, the isolation between these 2 ports is very less. This is what we saw in the last class, now if we want to improve the isolation between these 2 what do we need to do.

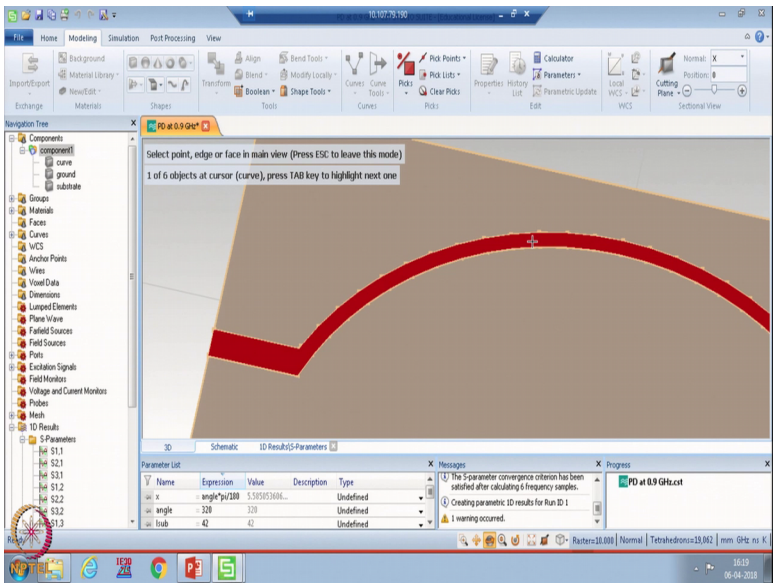
So, in order to do the isolation, if we go into theory or if you revise the previous lectures, we need to attach a resistor this should provide me the isolation. So, in case of 2 way equal power divider, we should attach a resistor between these 2 port, the value of the resistor should be 100 ohm. So, in this cases we are working at nine 100 megahertz we will use the SMD resistor and the size of the SMD resistor is very less. So, in our case we will be using 0603 SMD resistor. So, we should make the parts accordingly. Now, just to show you the results of 2 way power divider if you see here.

(Refer Slide Time: 02:24)



So, if you see S 12 and S 31 this is minus 3 however, if you see minus 3 2 this is minus 6 dB. So, the isolation is very less, now suppose if you have made this geometry and if you are not worried about the isolation between these 2 ports and if you just simply want to fabricate this particular PCB. So, what do you need to do you need to just simply select this.

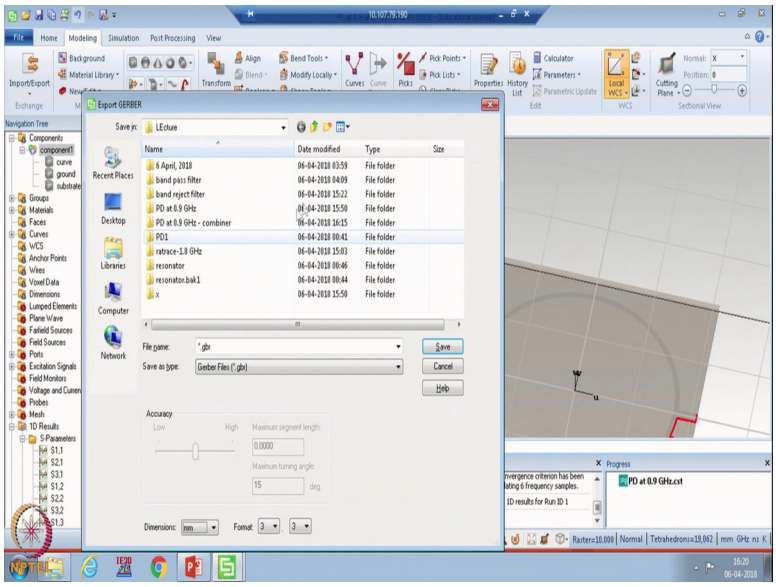
(Refer Slide Time: 02:53)



So, to select this, select the phase in which you have made the geometry then align your WCS with this phase. So, we have aligned here. Now if you want to fabricate this

particular PCB, all you need to do is you need to export it in Gerber format. So, I will show you how to export in Gerber format right now I have aligned my local coordinate system with the upper layer of this particular PCB. So, then if you go to export and then select 2 D option here you have a Gerber format. Now if you want to use other formats you can select the appropriate format. In most of the fabrication lab we need to give Gerber files. So, that is why we will be exporting it in Gerber format. So, just selected.

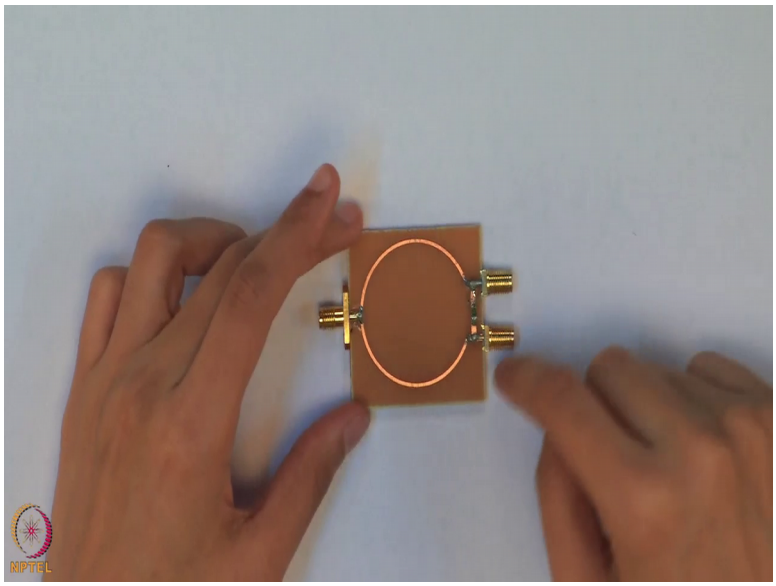
(Refer Slide Time: 03:45)



So, you can here select the appropriate folder, where you want to create the Gerber file of your geometry and you can name it as per your requirement. So, maybe I will just give the name as top. So, it will create the Gerber file, all you need to give is you need to just give that Gerber file to the manufacturer, and you need to tell about the substrate and the substrate size and you need to give the dimension and you need to tell them that it is a double sided PCB. So, he will fabricate your PCB.

So, this is regarding the fabrication process; now if you want to use this particular geometry as a combiner also. So, what you need to do? You need to provide a resistor between these 2 and to connect the resistor we should provide the parts accordingly. So, just to make the geometry of parts I will just select this layer, I will select this edge and then I will align WCS with this, and then I will make a strip between these 2 just to show you the geometry here this is a geometry of power combiner.

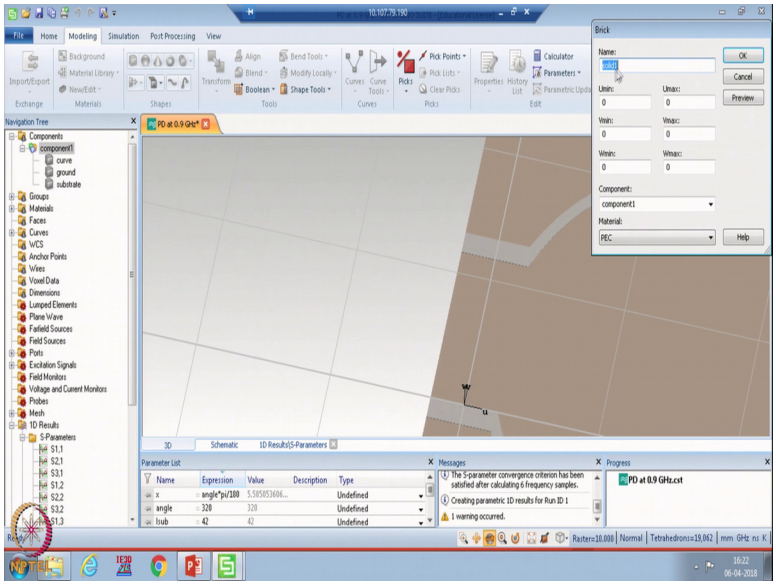
(Refer Slide Time: 04:56)



If you see here rest of the geometry whatever you see it is similar to the previous design, but in between you can see here one resistor is there. The size of this register is 0603.

So, 0603 resistor will be soldered on the pads. So, we need to make the pads to solder this particular resistor. So, here we will be making the pads. So, go to CST again and try to make these pads. So, we will simply create a brick.

(Refer Slide Time: 05:34)



So, to create a brick again go to brick and may be write it as pad, then in u here you need to just ensure that this line should be 50 ohm line. So, again the width of this should be corresponding to 50 ohm.

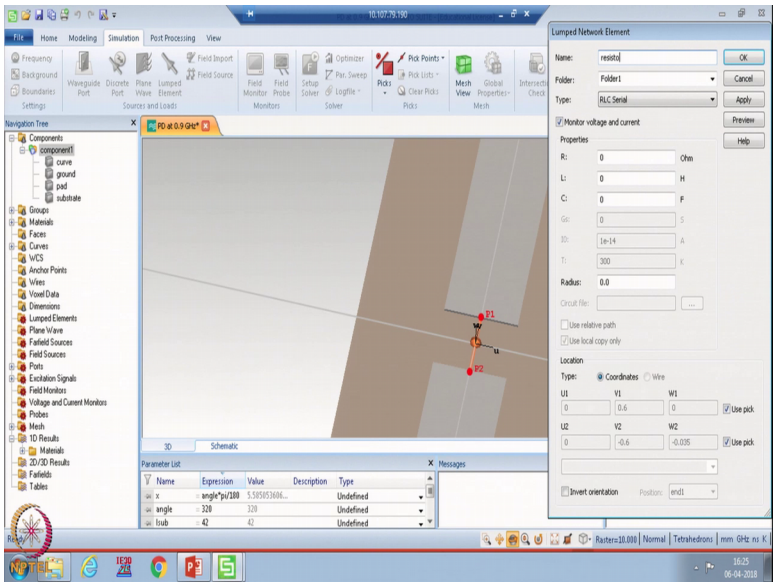
And so, V here will be nothing, but 2 into yeah. So, we have created one strip the thickness of this strip will be same as the copper thickness. So, t and the material will be PEC. So, now, we have created a pad. So, this is a strip now here there should be some portion which should be cut, because you want to place a resistor here and then you want to solder at these 2 ends.

So, at the centre just select the centre of this particular layer and then align WCS with that and then you cut a strip of this size, you select the size as per your resistor size. So, in our case this resistor that we will be using is 0603. So, for 0603 resistor the length is 1.6 mm and the width is 0.8 mm.

So, accordingly I will cut the patch from this microstrip pad. So, in U length will be same and then V may be just take the variable name lsmd by 2 sorry in this and for thickness may be minus t comma 0 yes. So, here may be just take it 1.6 or may be 1.2 whatever you want so, just to make the connection. So, we have created this step.

Now, we need to just cut this from the pad. So, you can see here just select this pad go to Boolean subtract, and then subtract this cut, now you can see you have created this space now what do you need to do? You need to just simply connect the resistor in between. So, just to connect the resistor you just select this edge.

(Refer Slide Time: 08:33)

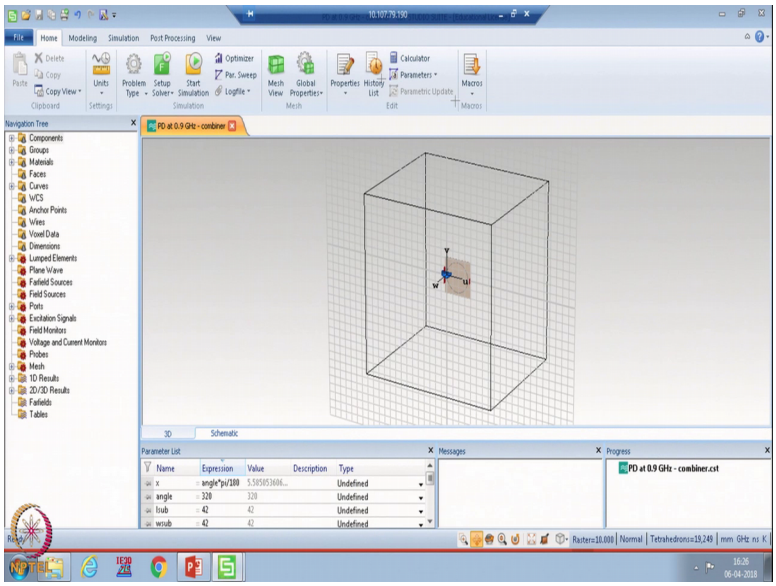


Similarly, you select second edge, and then go to simulation and select this lumped component option.

In the lumped component you can see here you can rain your rename your component as per your requirement. So, I will name it as resistor, and I will give the value as 100 ohm. So, now, you can see here if you go in to navigation tree one component of resistor has been created. So, now, my geometry is like this now we have created the geometry.

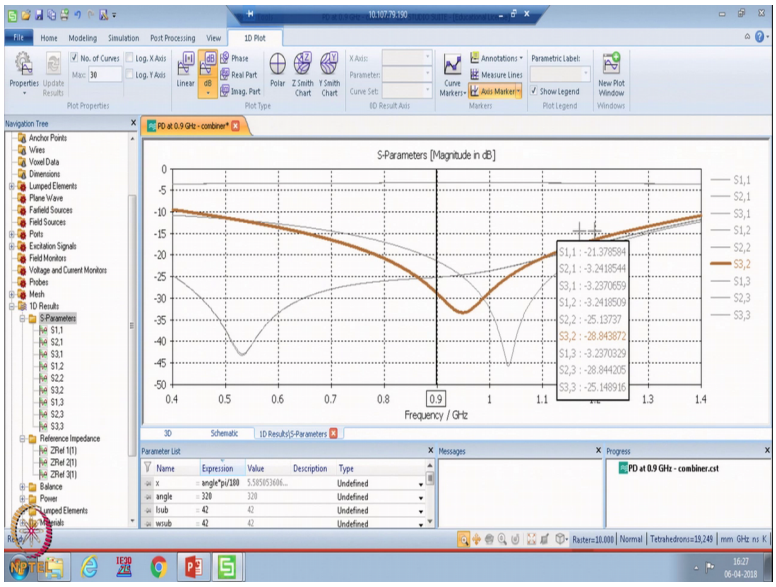
Now, we should see the performance and to see the performance just again start the simulation go to simulation or a then start frequency setup solver and then start the simulation in this case what we are expecting? We are expecting that the isolation between these 2 ports should be very high. So, the power from port to port through a port to port 3 should be very less. So, we will see results just wait for the simulation just wait for sometime yeah.

(Refer Slide Time: 09:46)



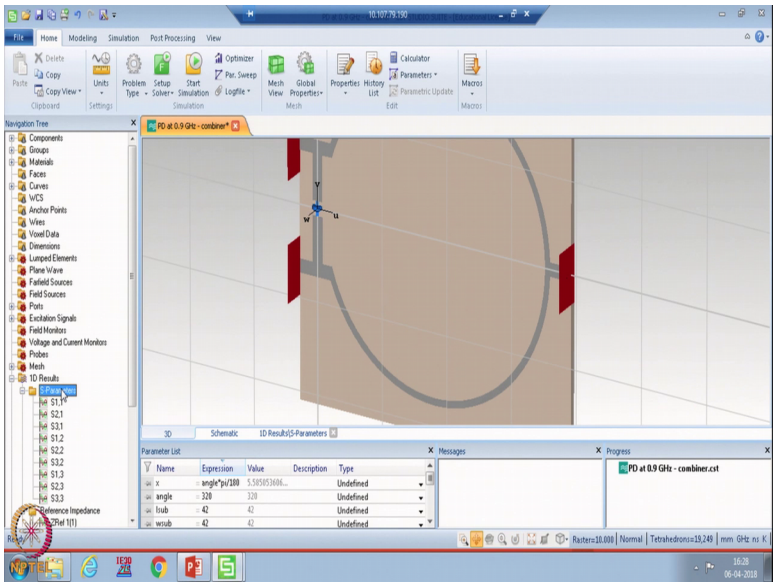
Now, if you see in this file.

(Refer Slide Time: 09:55)



So, in this particular case now if you see this is the geometry for the combiner this is a geometry corresponding to adamation.

(Refer Slide Time: 10:06)

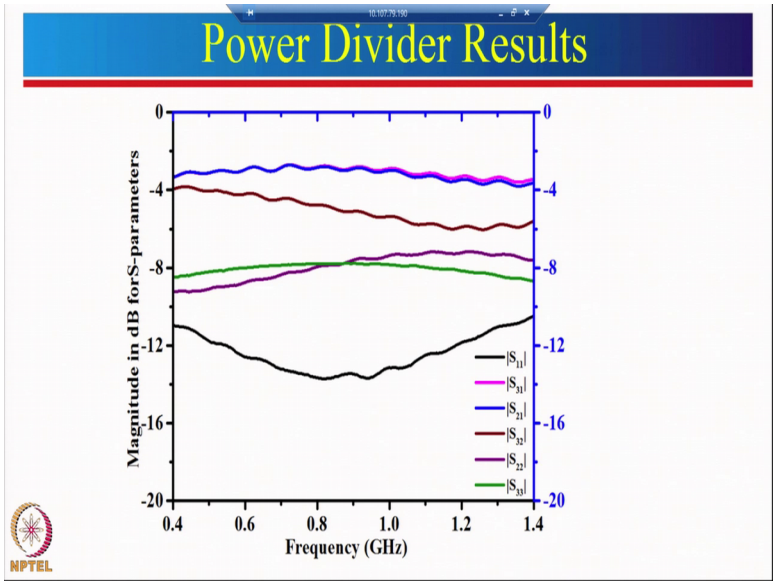


If you see the results simulated results here if you see this is my S11, this is S 12 that is now if you put the marker put here marker you see S 2 1 value is minus 3.24, S 31 value is minus 3.23. So, is approximately same as it was in case of 2 way power divider without resistor.

Now, if you try to observe S 3 2 see this value. So, this value is very less; however, in earlier case it was minus 6 dB. So, the isolation between port 2 and port 3 is very less now, if you give the input power at port 2 and port 3, then you can get the sum of these power at port 1. So, in this way you can design the power combiner. Now again if you want to fabricate this particular PCB use the similar procedure, just select this layer and then export it in Gerber form and then give it to manufacturer.

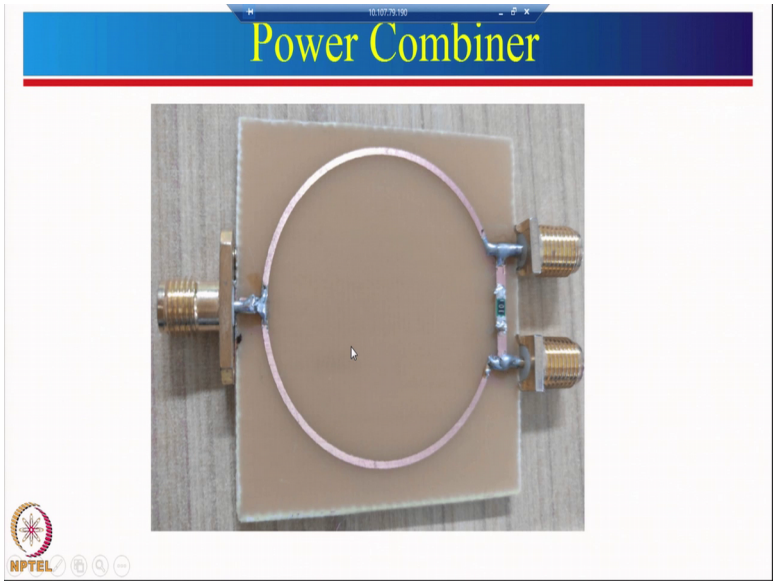
So, just to compare I have just fabricated these PCBs I showed you the PCB of these 2 power divider and combiners, now I will just show you the measured results of power divider in combiner, you see here in this is the geometry of 2 way power divider.

(Refer Slide Time: 11:32)



I measured these results here if you see this is S 21 and S 31. So, the power level is around minus 3 dB and; however, if you see S 23. So, this brown colour line is S 23. So, this level is very high because the isolation between those 2 ports; port 2 and 3 is very high. So, that is why this level is very high; however, if you see S 11 is just it is fairly matched in this particular frequency range.

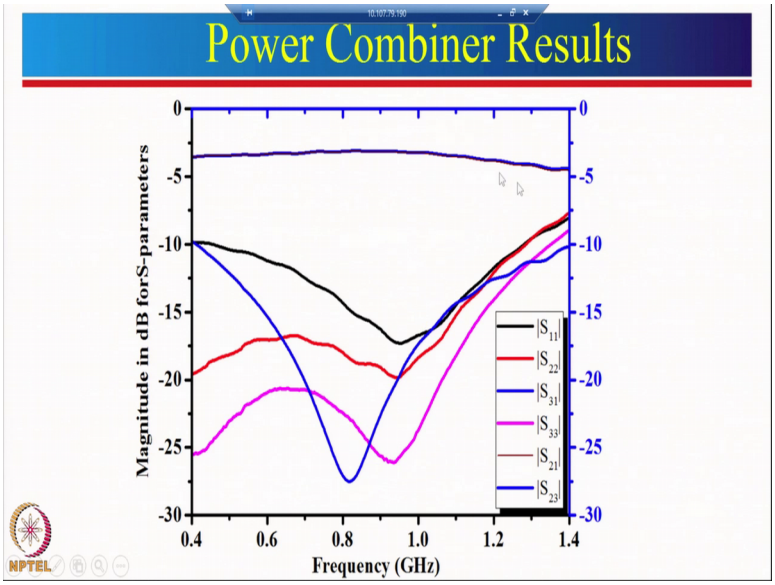
(Refer Slide Time: 12:03)



Then we made the power combiner; to make the power combiner we use the SMD resistor. So, this is the geometry for 2 way power combiner and then we measured this

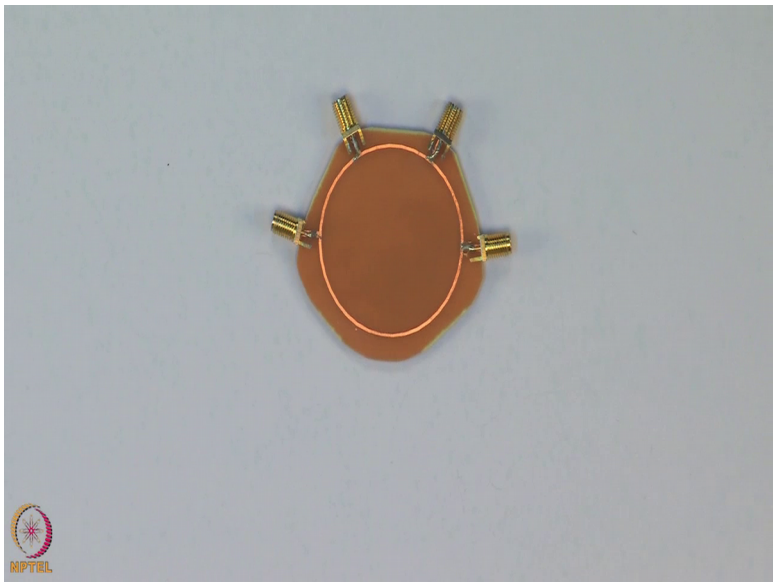
using network analyzer. So, in this case you need to have at least 2 port network analyzer, all you need to give is you need to give power at one port and you need to measure from other port. So, if you have 2 way power divider, you simply connect one cable here and another port over here and then terminate this with 50 ohm. If you have 3 way power divider then you give power from here and observe at these 2 ports.

(Refer Slide Time: 12:41)



So, if you see the measured results of this power combiner, here you can see in this particular case the S 31 and S 21 is minus 3 dB and S 2 3 that is the isolation between port 2 and 3 is very good as compared to the previous case. It is below minus 15 dB our desired range, even you can say below minus 20 dB also. So, in this way we designed power combiner. In the next lecture we will try to design hybrid coupler.

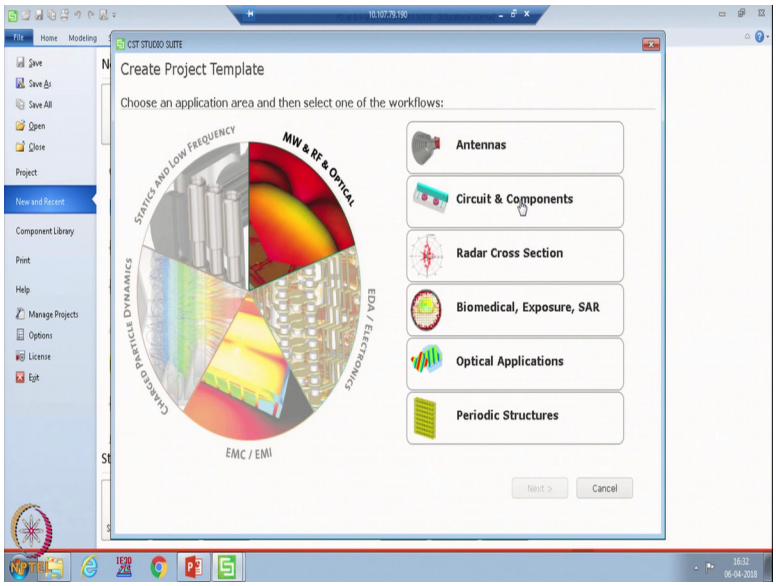
(Refer Slide Time: 13:21)



Now, we will try to design hybrid coupler that is rat race coupler, we will try to design the rat race coupler at 1.8 gigahertz. So, this is the geometry of rat race coupler it is also designed on FR4 substrate for 0.8 mm thickness. So, here this is rat race coupler, the all these sections are lambda by 4 sections and this section is 3 lambda by 4 sections. Now if you recall the basic theory of rat race couplers.

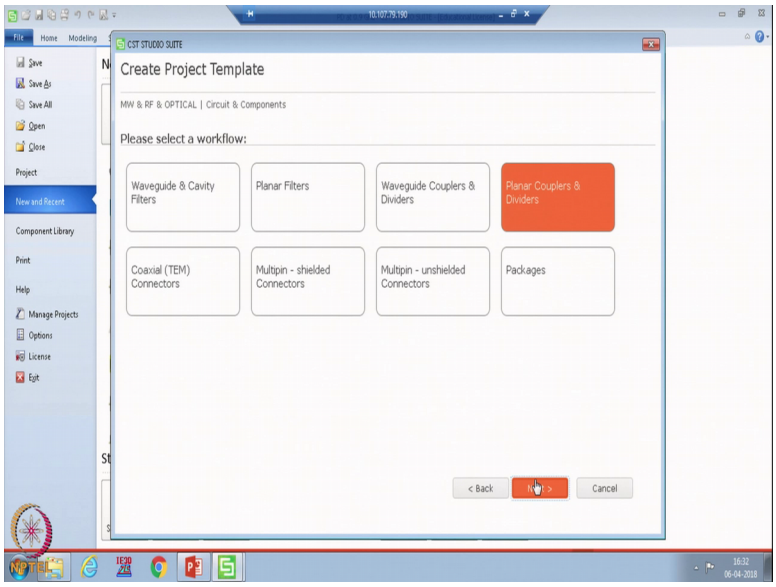
So, this line length is 3 lambda by 2, if you see the electrical separation between these. So, this is lambda by 4 lambda by 4 lambda by 4. So, this is 90 degree; however, if you see special separation. So, they are separated by 60 degree. Now, we will try to design this hybrid coupler in a CST microwave studio. So, let us start again CST create a new file.

(Refer Slide Time: 14:18)



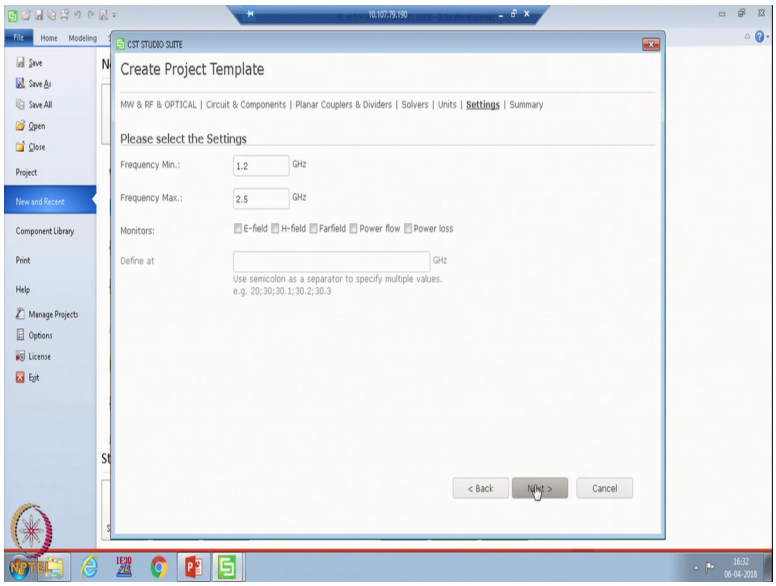
And repeat the same procedure.

(Refer Slide Time: 14:25)



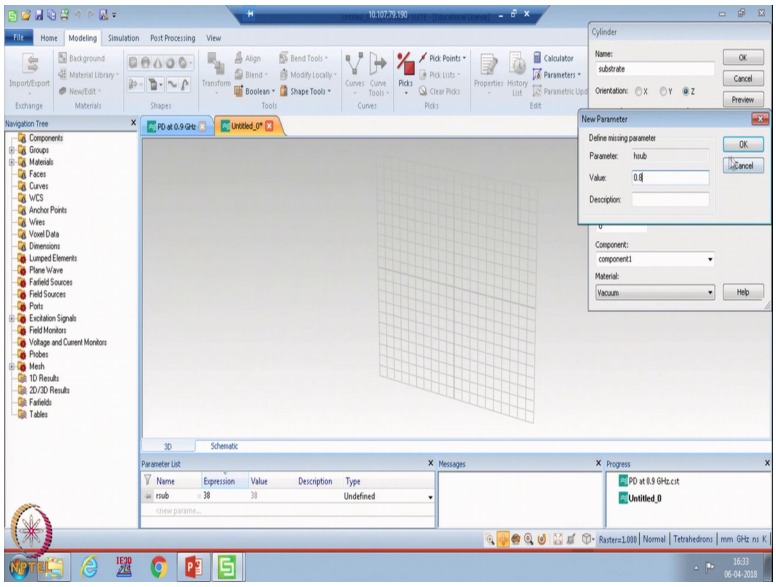
So, this hybrid coupler we are designing at 1.8 gigahertz.

(Refer Slide Time: 14:33)



So, maybe take the frequency range from 1.2 to 2.5 gigahertz.

(Refer Slide Time: 14:48)



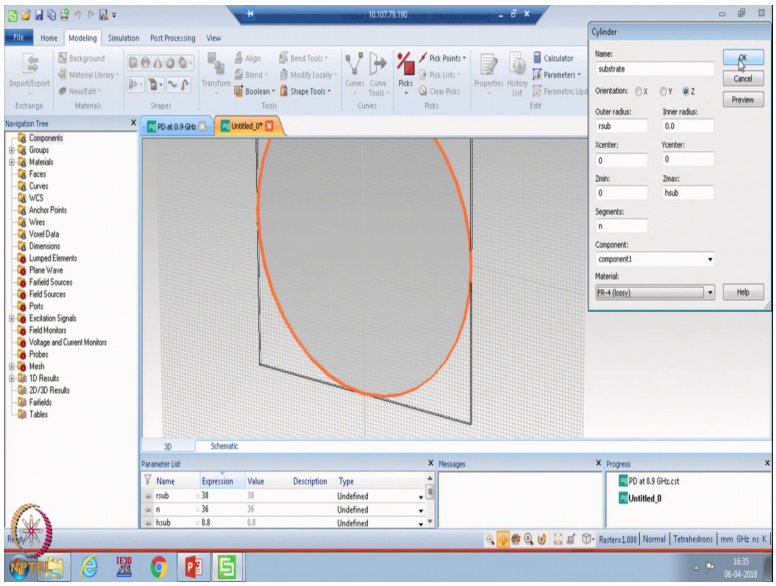
So, in order to design this, since it is a circular geometry we will take the substrate of circular geometry only. So, to design circular substrate go to in modeling, go to cylinder and then escape maybe name it as again substrate write it as r sub then z is hsub. Take rsub is 38 mm, hsub is 0.8 mm.

(Refer Slide Time: 15:40)

So, just to tell you here the dimensions that we are taking for this hybrid we are taking FR4 substrate of 0.8 mm thickness, we are taking the inner radius is 22.4 mm and outer radius is 23.2 mm. These things you can easily calculate using the line calculator you know this electrical length and you know this periphery that is equal to 2 pi r an electrical length is 3 lambda by 2. So, that will be equivalent to 540 degree.

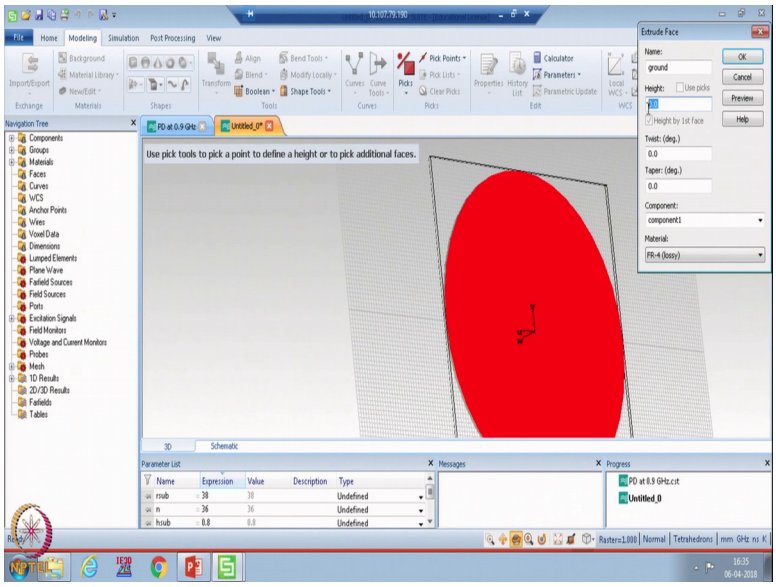
So, you can use simple line calculator and from there you can calculate the inner and outer radius. Here the strip width is 0.8 mm that corresponds to 70.7 ohm how does this come for that you can refer the theory of rat race coupler this particular impedance calculation was told to you in the previous lectures. So, we will just simulate this geometry here. So, now, if you see here this has created the circular section.

(Refer Slide Time: 16:42)



Here you can define the number of segments, I will take it as may be variable again we will define the number of segments let us take it as 36 and the substrate again FR4 substrate selected from the library option; load. So, this is FR4 substrate.

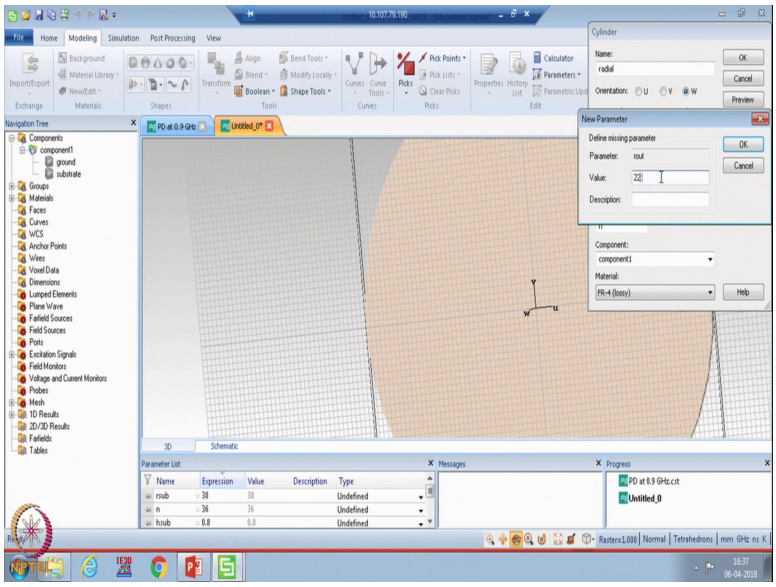
(Refer Slide Time: 17:07)



So, we have created the FR4 substrate. Now to make the radial circular strip enable first local coordinate system align it with the top face and on the back side as I told you we need to make the ground. So, you can make the ground just to do that select the other face and again use this that extrude option, and then name it is ground and give the

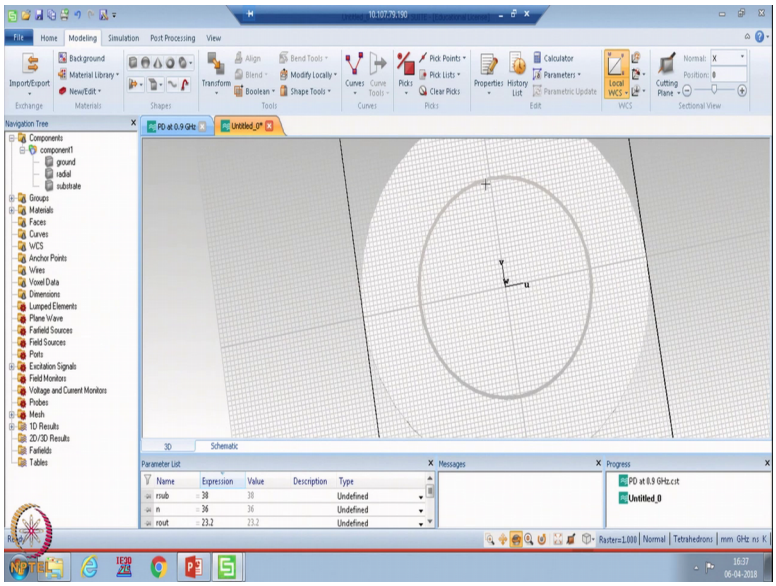
thickness as you use it for copper that is 35 micron. So, far we have made the ground and the copper. So, for the ground the substrate should be chosen as PEC substrate. So, mistakenly if you select any other substrate you can change it from this option.

(Refer Slide Time: 18:13)



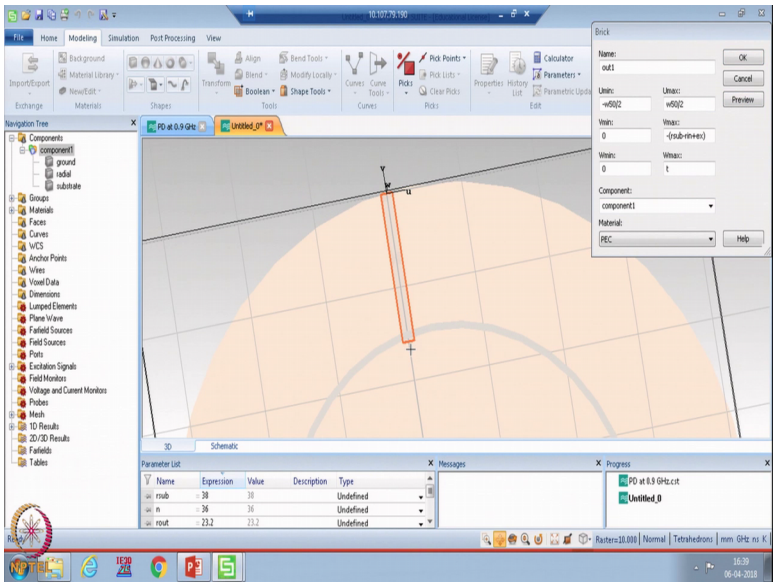
Now on the top side again you draw a radial strip, for radial strip what do you need to do? You need to just again go to cylinder escape radial then you name it as rout then rin 0 t and again the number of segments keep it is n that is 36. So, rout take it as 23.2 I do not recall just to recall I think its 22.4 and 23.4. So, this is rout is 23.2 and rin is 22.4. So, substrate for this is PEC. So, now, we have created the radial strip.

(Refer Slide Time: 19:15)



Now, to make port all you need to do is. Just go to the option yes.

(Refer Slide Time: 19:26)

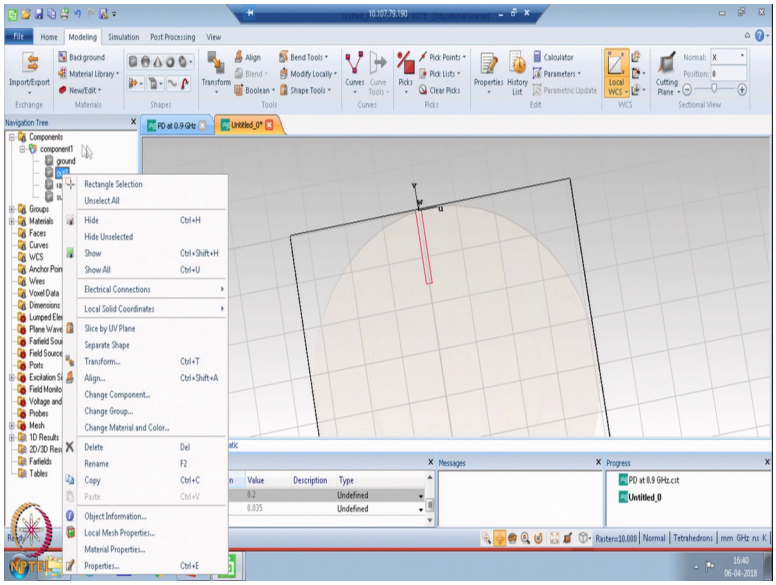


So, select this particular segment substrate. Select a segment to select a segment select a particular edge may be this one, align your WCS with edge now you need to define a port. So, to define a port again go to brick and then may be in name it as out one, and this port should come up to this particular point.

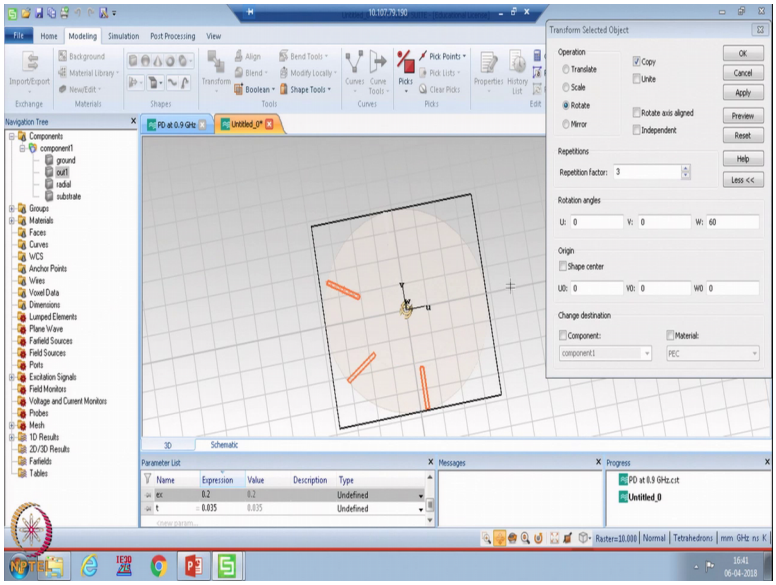
So, to define it you know this separation is how much this is rsub minus rin. So, we will take it accordingly. So, in V coordinates what you should give rsub minus rin plus some

extra length just to ensure that there is a connection between radial strip and the output port and thickness t. And in U this should be minus w 50 by 2 w 50 by 2 and w 50 will be 1.55 and ex maybe take 0.3. So, here this you need to take in negative because positive is on the other side this should be rsub yes you take this is 0.2. Now, what do you need to do you need to just.

(Refer Slide Time: 21:58)



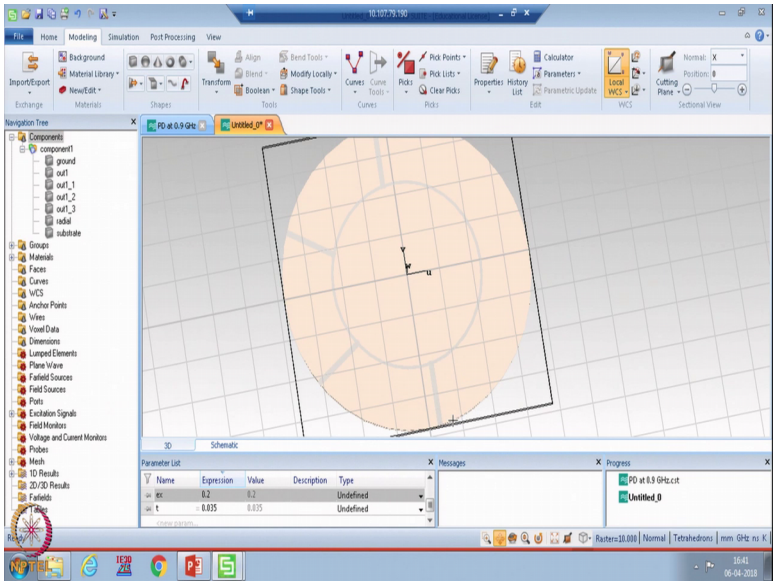
(Refer Slide Time: 22:02)



First with this particular face align your local coordinate system and then you rotate these faces by 60 degree. So, to rotate it select this out transform then rotate and copy

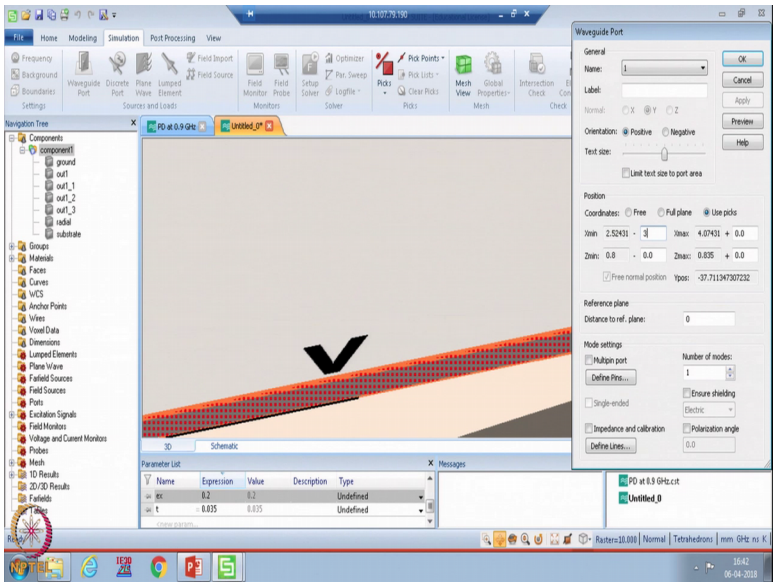
here you give 60 degree and since you want to make 3 other ports. So, you can increase the repetition factor you can see here right. So, you see we have created 4 ports.

(Refer Slide Time: 22:42)



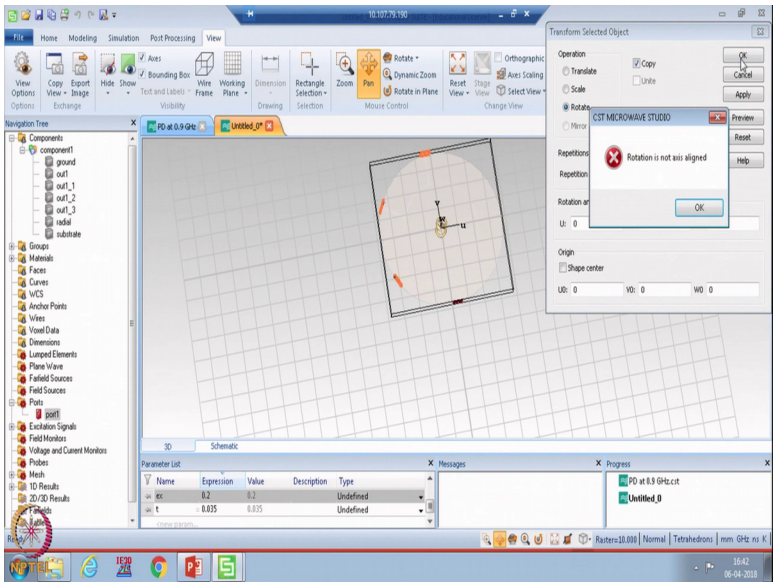
So, this is a geometry corresponding to our requirement. Next we need to make the ports. So, to make ports all you need to do is again replicate the same procedure, select this particular edge yes then define waveguide port and use the same procedure as we told you earlier 3 into hsub and this 4 into h substrate.

(Refer Slide Time: 23:05)

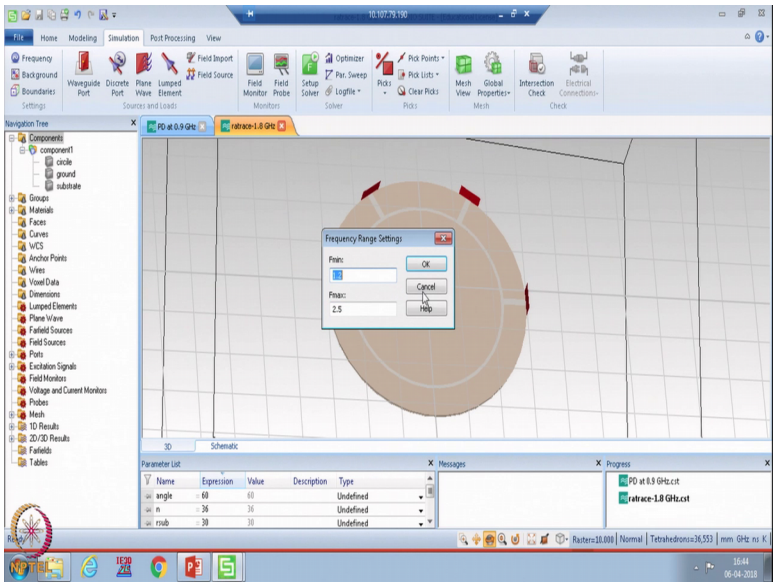


So, this is how you create a port and then you can rotate this port again. So, transform to rotate it; select rotate, select minus 60 and copy you see you can simply create these number of ports if you see this is a geometry.

(Refer Slide Time: 24:01)



(Refer Slide Time: 24:05)

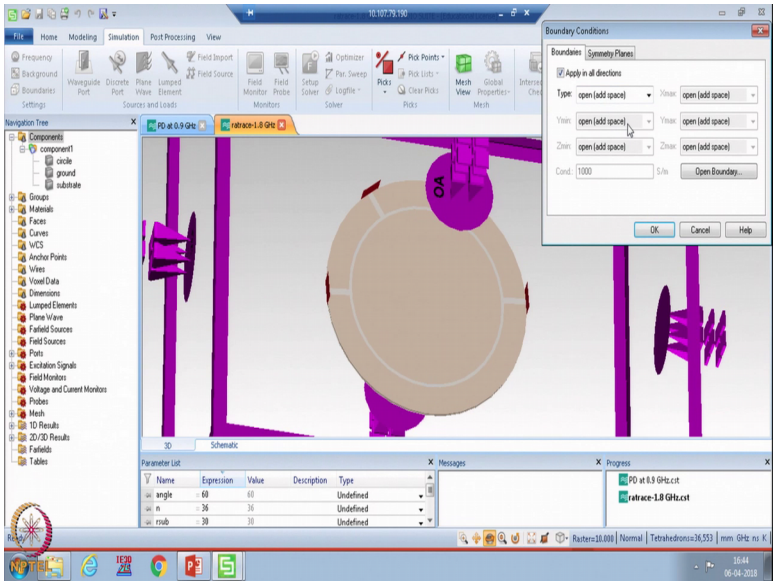


Now, we have created all the ports 1 2 3 4 now we need to simulate.

So, to simulate it again you need to give all the background properties boundary conditions and the frequency range for that, go to simulation define frequency range

since we have already given the frequency range we need not to define it again then you select the boundary properties open add space.

(Refer Slide Time: 24:31)



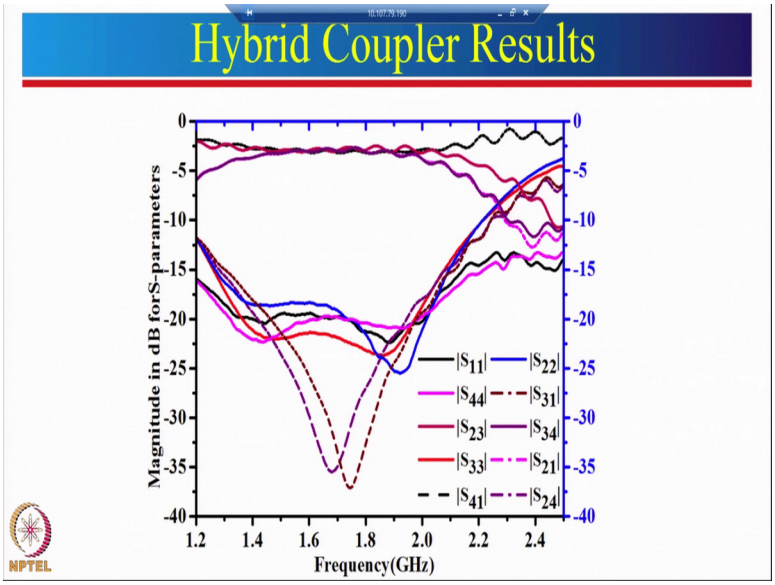
If that is already not unable you need to apply it in all the directions and then start frequency setup solver start the simulation and then wait for the results. So, it just take some time. So, mean while we will discuss what we are expecting from our simulation.

So, now if you give input at this particular port, this length will be pi lambda by 4 and this will be lambda by 4. So, power here will be in same phase, now if you see at this particular port this length is lambda by 2 and this length is lambda. So, they are in phase reversal. So, the phase difference between these 2 is 180 degree. So, here at this particular port the power will be in opposite phase. So, it will cancel out. So, ideally the power at this port should be 0.

Now, if you see at this particular port this length is 3 lambda by 4, again this length is 3 lambda by 4. So, again here the power should be half and it should be in same phase. Now, if you compare these 2 ports. So, here it will be shifted by 90 degree and in this case it will be shifted by minus 270 degree. So, we will try to observe this in the our simulation result now just go to simulation result so, the simulation is still going on. So, in between I will just show again if you want to fabricate this particular design you export the geometry in Gerber format.

You see this is the fabricated PCB, now again just to measure it you give input at one port, if you have 2 port network analyzer, then you connect the other port at any of these outputs and other 2 port should be terminated with 50 ohm.

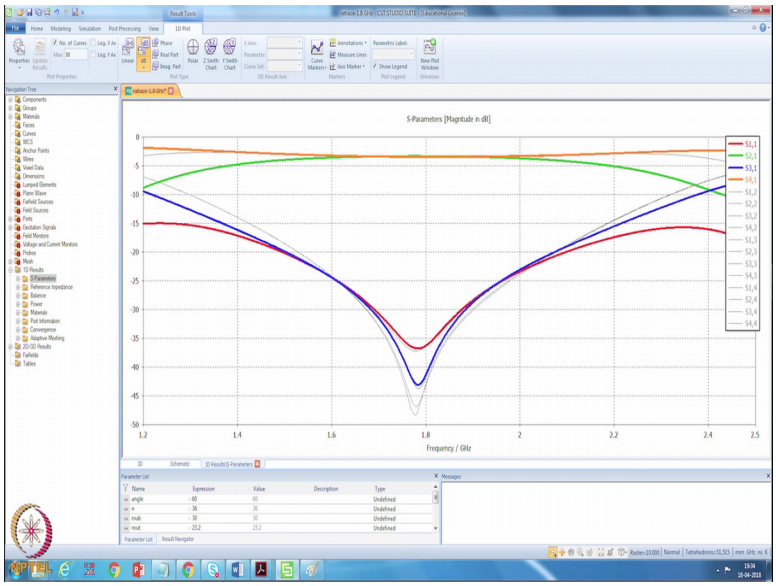
(Refer Slide Time: 26:30)



And in the similar way you can do the measurement for all the ports. So, these are the measured results for this geometry, you can see here this is S 11 black one, this line is S 11 this blue one is S 22 and this red curve is S 33. So, this was designed for 1.8 gigahertz and for S 13 as I told you that is isolated ports you can see S13the isolation is very high at this particular port. So, the power going to that particular port is very less.

Similarly, if you feed at port 4 that is port 4 let us take may be this as port 4, then the (Refer Time: 27:12) at port 2 will be very low. So, you can see S 2 4 that is this one this is very low. So, in this way you can design a hybrid coupler just to see the simulated results here these are the simulated results.

(Refer Slide Time: 27:29)



In the red line you can see that this is the S 11 curve, the green one is the S 2 1 curve. So, at port 2 the power should be minus 3 dB similarly at port 4 power should be minus 3 dB. So, S 21and S 41 both are minus 3 dB wherever S 31 is very less. So, it should be will be low minus 20 or minus 30 dB which we will show that this port is isolated. So, in this way we will simulate the hybrid coupler.

So, in this particular lecture we tried to design the power combiner and then we tried to put the SMD resistor component between the 2 ports and then we saw the performance of power combiner. And we saw that the power from port 2 to port 3 is very less. So, these 2 ports if we feed the power from the 2 output ports of power divider, then the power can be combined and the combined power can be observed at the port 1. After that we tried to design the hybrid coupler 4 way rat race hybrid coupler and then we saw the performance if we feed power from one port what will be the power at other port.

Now, just to highlight one more thing if we give power from port 1 and port 3 by using a same concept here we should get the sum of these power and here we should get the difference of these power. So, by using the same concept you can do the simulation. Just to excite these ports you need to just give power here and at this particular port and just give the amplitude 0 at these 2 ports and you can easily verify the concept of hybrid coupler. So, in the next lecture we will try to do the simulation of active circuits using CST microwave studio.

So, for this lecture thank you very much, we will see you in the next lecture bye.