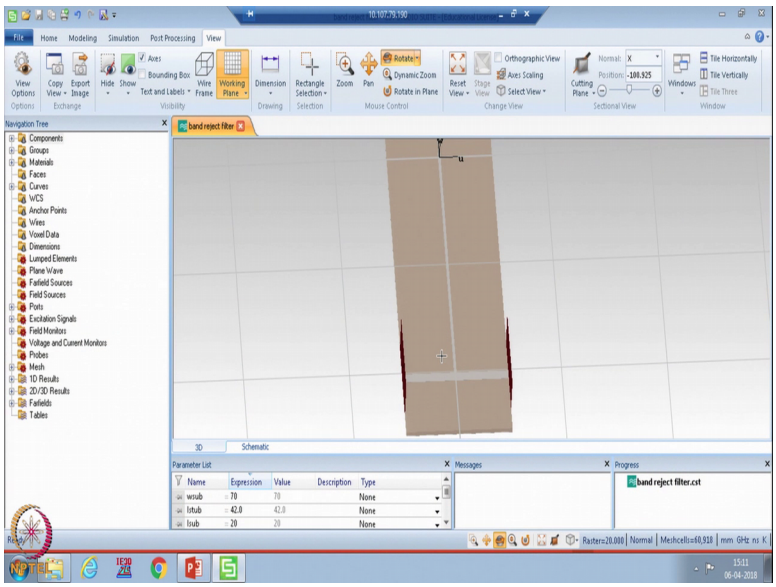
Microwave Theory and Techniques
Prof. Girish Kumar
Electrical Engineering Department
Indian Institute of Technology, Bombay

Module – 12
Lecture – 56
Power Divider and Combiner Design in CST

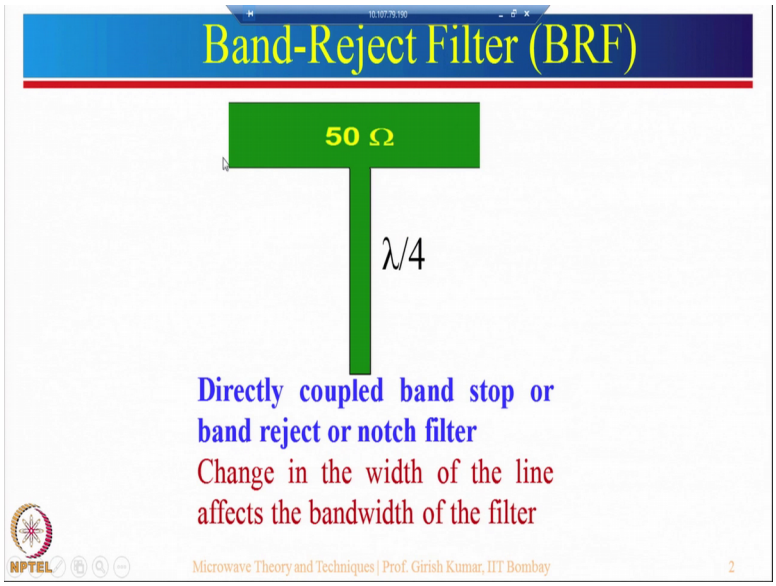
Hello, in the last lecture, we discussed about CST microwave studio. Firstly, we introduce the interface of CST microwave studio. We started with different modeling shape, we tried various shapes using CST microwave studio modeling option. After that, we saw different solvers, which we can use using CST microwave studio. And we tried various features of CST microwave studio. After that, we tried to make microstrip filter. So, we started with band reject microstrip filter, and we tried to model microstrip band reject filter. So, let us continue.

(Refer Slide Time: 01:05)



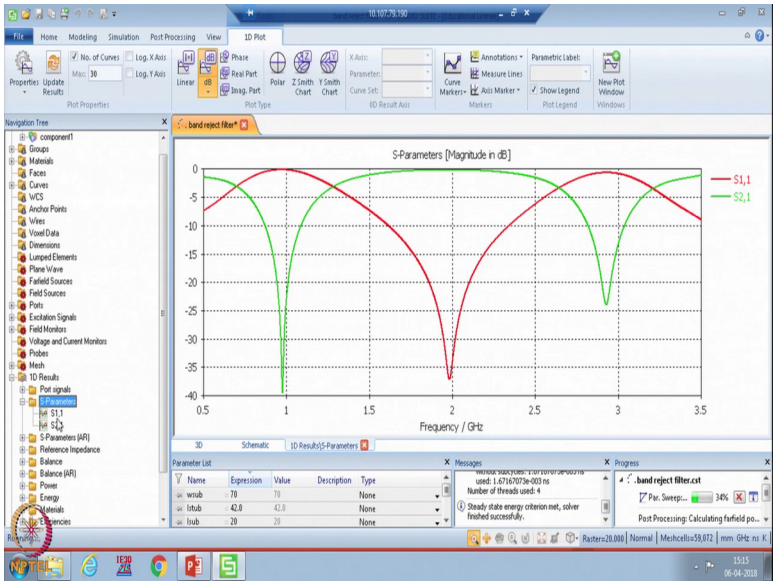
So, we made a microstrip band reject filter.

(Refer Slide Time: 01:10)



Just to show the concept, this is a band reject filter. Here we use the input and output microstrip line just to give the input, and measure output signal and it is stepped by a lambda by 4 microstrip line, which is opened. So, if it is open at this end, it will act like a short circuit because of quarter wave transformer behavior. So, we modeled this particular microstrip band reject filter in CST microwave studio. So, let us open the CST microwave file. Here we made this strip line to measure output and give input. And then, we used in lambda by 4 microstrip line of 0.5 mm width and 42 mm length. And then, we tried to simulate this particular filter.

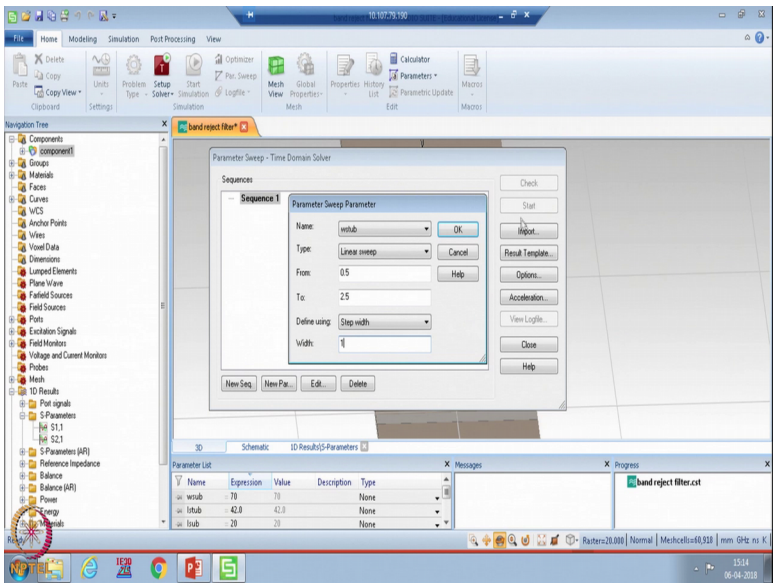
(Refer Slide Time: 01:59)



After simulation, we received these results. Now, if you see here, you can seen from this particular plot is S2, 1 is showing, band reject behavior at 1 gigahertz, and at 2 gigahertz. It is showing band pass filter, and at 3 gigahertz, it is showing again band reject filter. So, at 1 gigahertz, we made a lambda by 4 microstrip line. So, it was showing the band reject filter. Now, if you see at 3 gigahertz, it is thrice of the 1 gigahertz. So, the length corresponding to this frequency will be 3 lambda by 4. So, again it will show band reject filter.

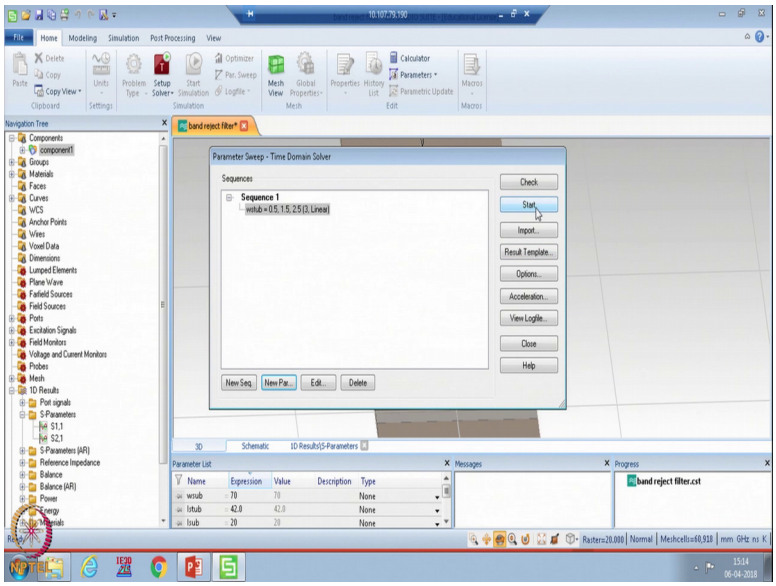
However, if you see around 2 gigahertz, the length will be around lambda by 2. So, an open will act like a open. So, in that case, it will act like a band pass filter. Now, I just want to tell about the other features. Here, as I told you in that parameter window, we have used various parameters using variables. Now, we can change various parameters using this parameter sweep options.

(Refer Slide Time: 03:06)



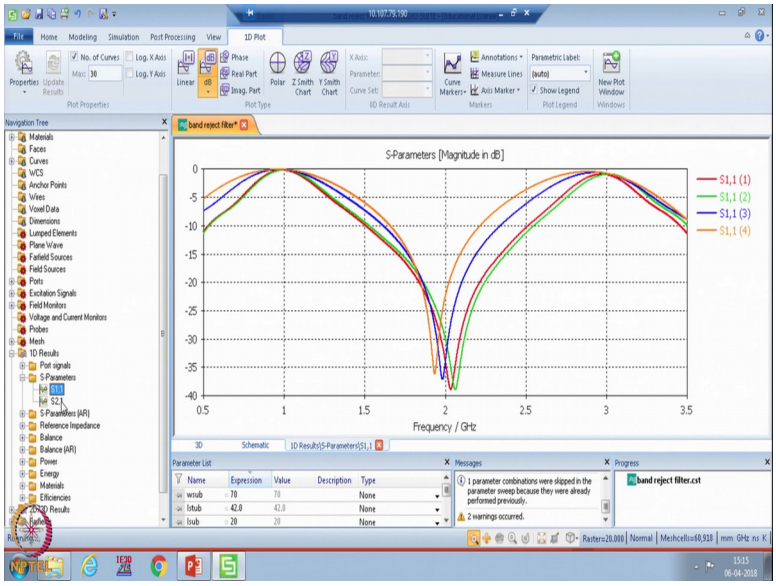
Now, go to parameter sweep, select first new sequence, then you select new parameter. Now, I want to change the width of the stub, because I want to see the behavior of the stub width. So, I will select wstub, and this range is 0.5. So, just to see, I will change this width from zero point to 2.5 in the step of 1 mm. So, I have given here 1 mm step width, and then press.

(Refer Slide Time: 03:36)



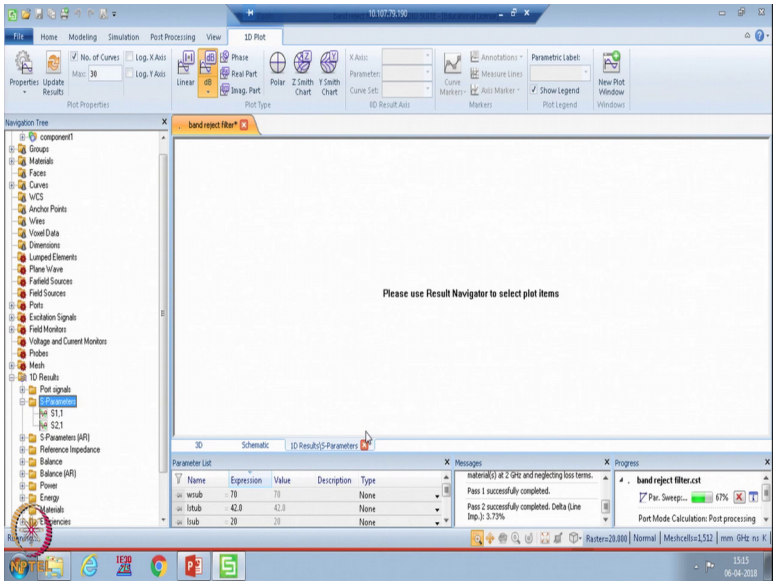
Now, you see here, three samples are created. One is 0.5, 1.5, 2.5, because I took the step width of 1 mm. You can take the width as per your requirement, and then you can do the analysis for all the values. Now, just start this parameter sweep. It will just take some time. So, in this way if you want to optimize your particular design, and if you are not sure about the final parameters, then you can use the parameter sweep option. It will show you the results corresponding various parameters, which you have chosen.

(Refer Slide Time: 04:21)



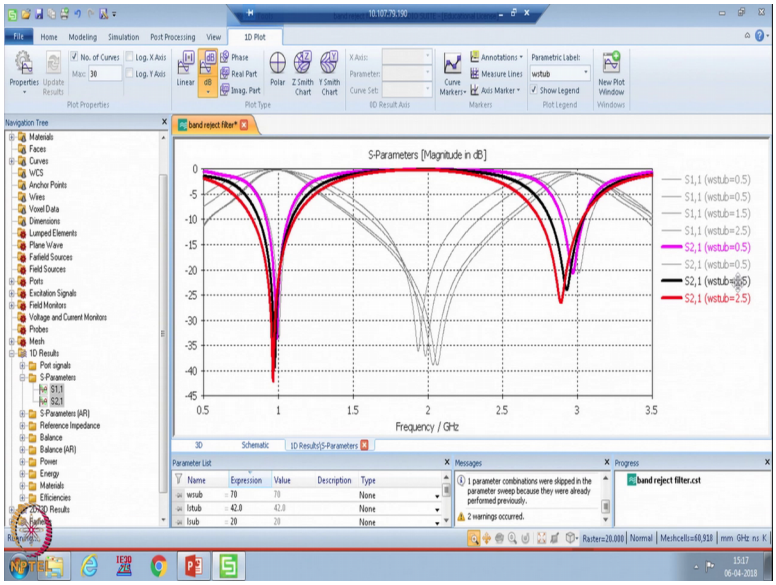
Right now, I have just changed this from 0.5 to 1.5. So, you can see here, it will show the results corresponding to all the design parameters.

(Refer Slide Time: 04:25)



So, right now the simulation is going on just wait for few seconds, then it will show the results. Now, it has already stimulated, now just see the results.

(Refer Slide Time: 04:41)



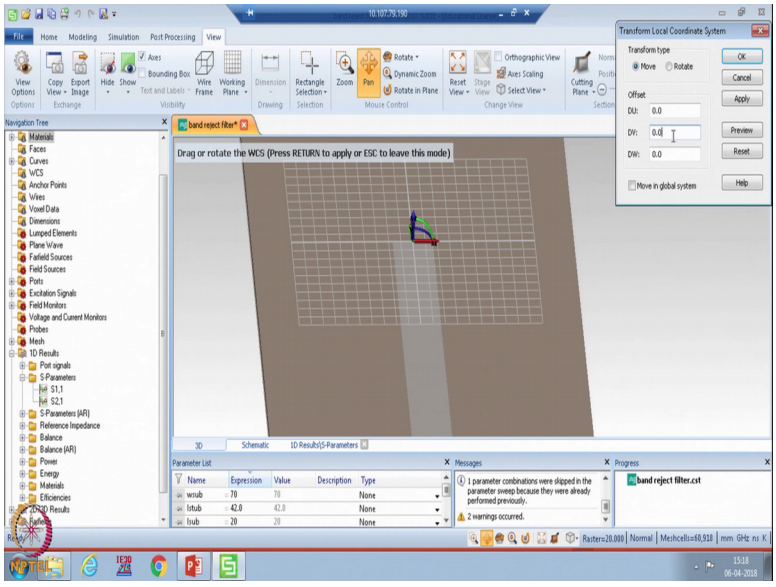
Now, if you see, so this is the result corresponding to our 3 simulation, which we did. Now, here you see various curves are there, and you very different values is these curves are corresponding to those values. Now, if you want to see which value, corresponds to

which curve. So, to check that go to parametric levels, select your parameters, which you have used for parametric study. So, we selected wstub, so I will just select here wstub. Now, you can see here, this first curve, this one is corresponding to wstub equal to 0.5 mm, then this one is corresponding to 1.5 mm, and this one is corresponding to 2.5 mm.

So, now you see these curves simultaneously, you can see here by changing the stub width. Let me just select for your reference yeah. So, now I have selected these 3 curve. One is corresponding to 0.5, another one is corresponding to 1.5, and this red one is corresponding to 2.5. Now, you can see varying this stub width, the band reject bandwidth is changing. So, in that way, you can design the filter as per your requirement. So, this feature is in CST microwave studio.

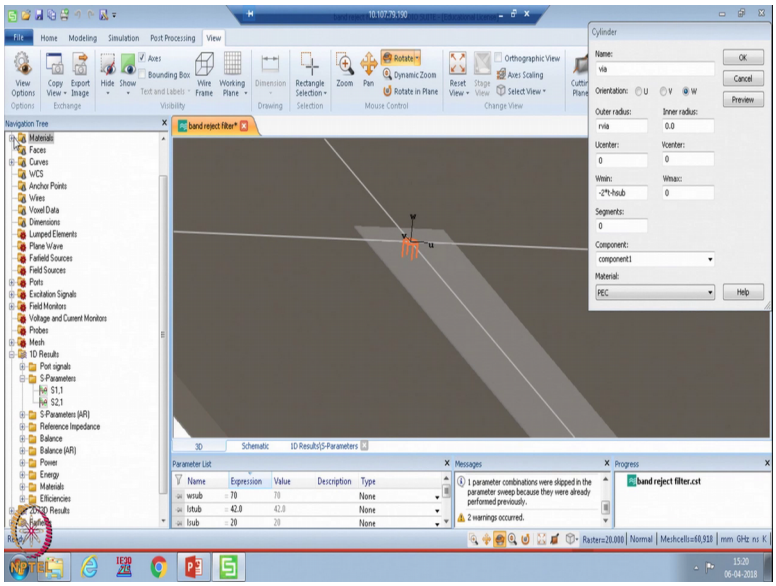
Now, next I want to tell you, how this band reject filter can be used as a band pass filter. So, if you see the concept part in this, suppose if I put a short here, if I put a via and short it with ground, then it will act like a short. And now, this short will act like a open at this end. So, it will pass, all the frequency points corresponding to lambda by 4. So, then it will act like a band pass filters.

(Refer Slide Time: 07:09)



Now, we will try to design band pass filter. So, to do that just go to modeling, select this edge align WCS. And then just give a offset, because you want to put a via over here. So, give some offset for local coordinate system. So, use transform, and now so you need to transform in V by some x parameter. Let us take x as around maybe 1 mm.

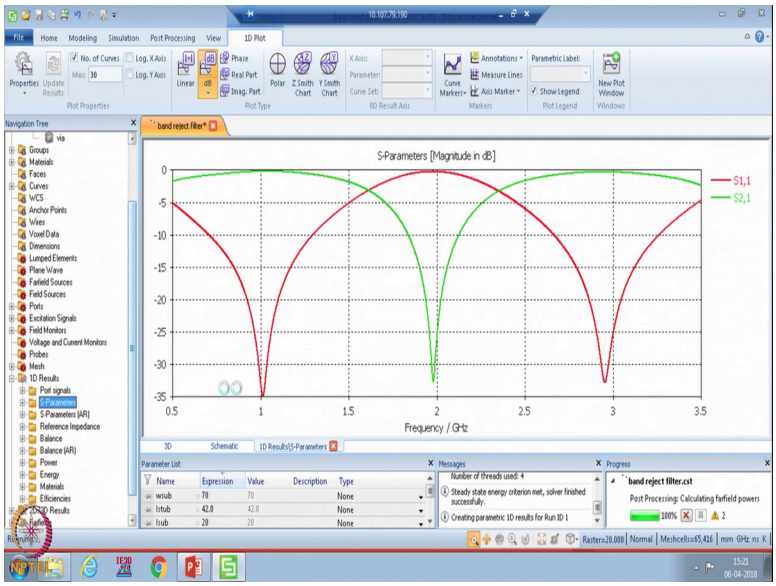
(Refer Slide Time: 07:31)



Now, you try to make a via so, via will be a cylinder between the ground plane and the upper strip. So, it will make a electrical connection in that way, it will act like a short. So, now we will try to make the cylinder. So, go to cylinder, select that option, name it as via maybe give outer radius as maybe rvia. And then, you define the W coordinates, because W is representing the thickness in this case, so you define W coordinates. Center we have selected 0, 0, because we align the local coordinate system, it is center point of via only. So, we need not to give the absolute value of this, it will be 0, 0 in this case.

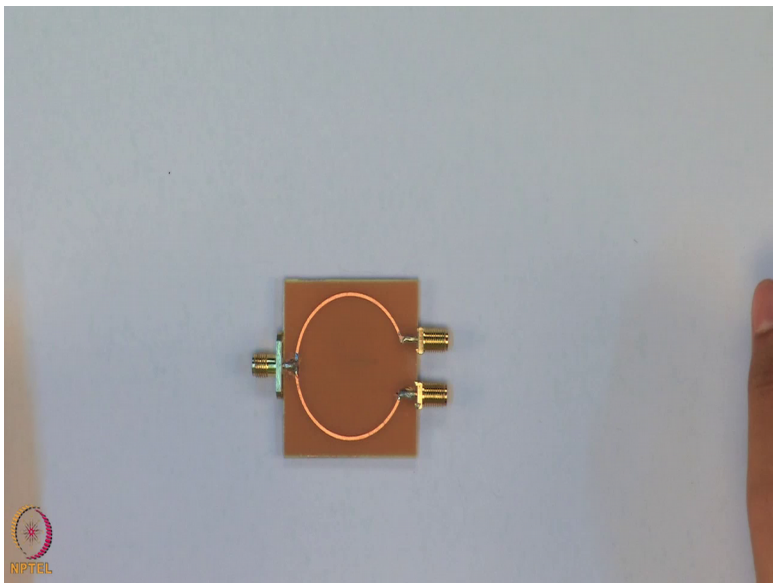
So, our local coordinate system is aligned and with the upper strip so, W maximum will be 0, 0 whereas W minimum will be equivalent to minus 2 times of t minus h substrate. So, now if you see here from geometry itself, it has created a cylinder. Just to show you, I will just then press ok. And just to show the proper connection, I will just hide the substrate. And now, if you see here, this via is making connection between ground plane and the upper strip. So, in this way we have created a short. Now, just to see its behavior, we will simulate it again simulate. Now, you try to observe its behavior. So, let us see the response yeah.

(Refer Slide Time: 09:23)



So, now if you see the response, just try to observe S2, 1, because S2, 1 is the measure of how much power is going from port 1 to port 2. This is port 1, and this is port 2. So, how much power is going from port 1 to port 2. Now, if you see here, this is S2, 1 green curve, and this red curve is S1, 1. So, if you see S2, 1 response, so it is acting like a band pass filter at 1 gigahertz. However, earlier it was acting like a band reject filter. So, just by placing a short at the end of stub, it is behaving like a band pass filter. So, in this way we can design band pass filter or band reject filter using microstrip line easily.

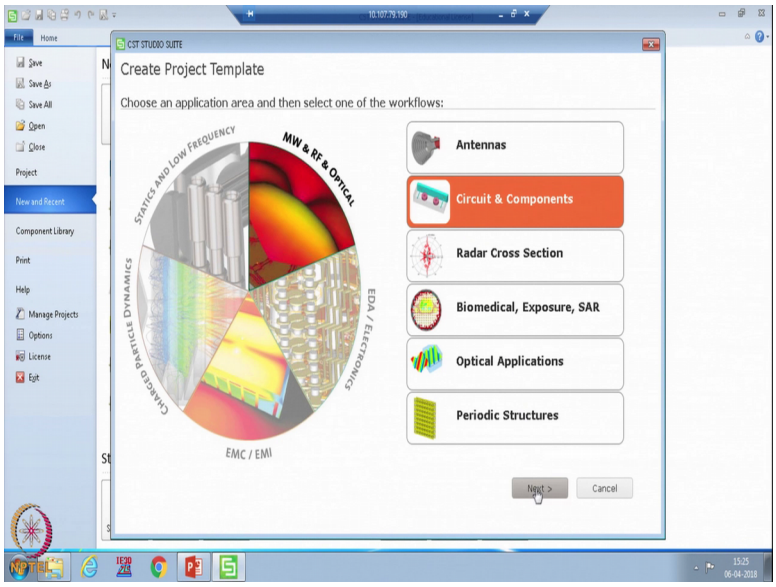
(Refer Slide Time: 10:18)



Now, we will try to design a two way power divider. So, this is a PCB for two way power divider, it is made on FR-4 substrate, whose thickness is 0.8 mm. Here, this is input port, and these two are output ports. So, you can name it as port 1, 2 and 3. So, if a power of 1 milli watt is given at port 1, you will get half of the power at port 2, and half of the power at port 3.

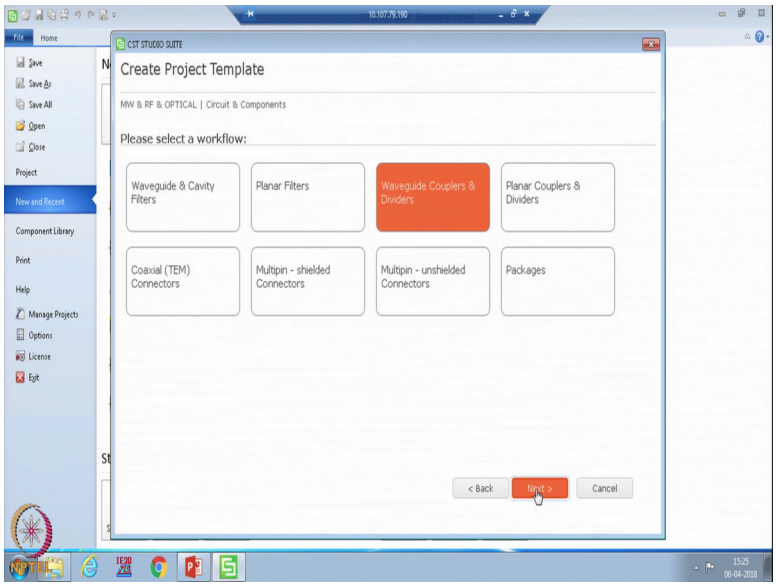
So, just to show you the design concept this strip is a 70.7 ohm impedance strip and its length is lambda by 4. It is just the basic concept of two way power divider the same is in the case of this particular strip. So, now we will try to make this particular power divider. On the back side if you see, it is full ground plane. So, how we will start, we will first make the substrate. And then on the bottom side, we will make full copper that is ground plane. And on the top, we will try to make this type of design.

(Refer Slide Time: 11:32)



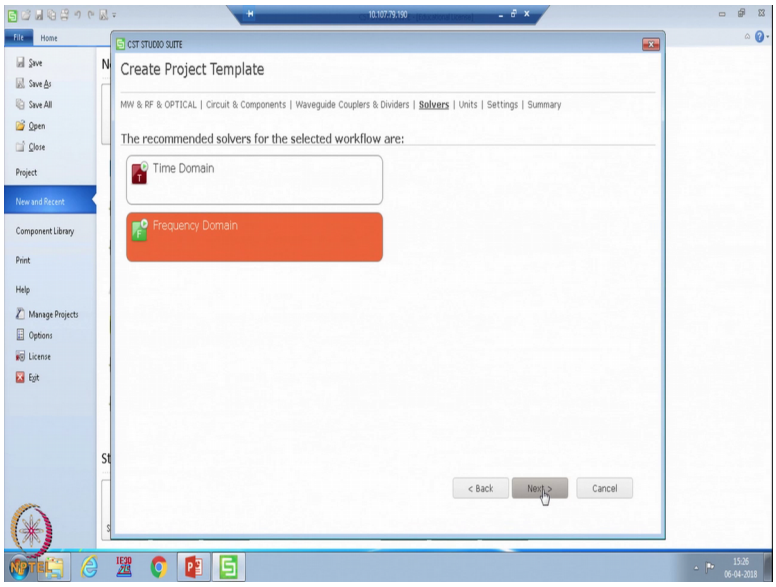
So, let us start with this design again open CST microwave studio, then create a new file. And just repeat the same options that I told you in the last class, select the appropriate component.

(Refer Slide Time: 11:39)



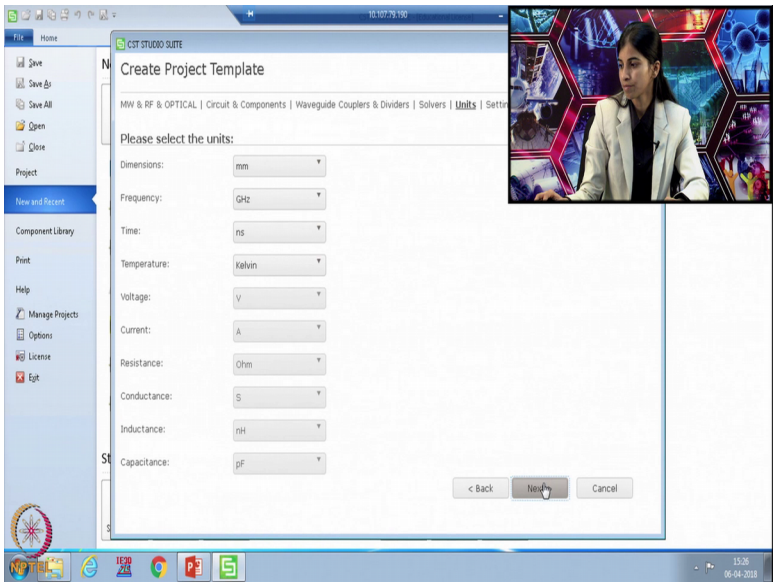
And then, select the appropriate module where you want to work.

(Refer Slide Time: 11:48)



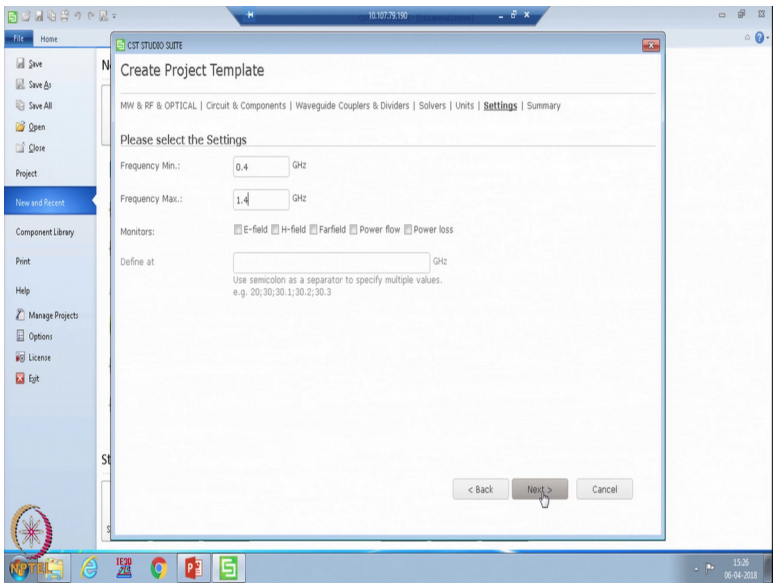
And this time, we will do the simulation in frequency domain. Last time, I did the simulation using time domain method, this time we will use frequency domain method. So, I have selected frequency domain.

(Refer Slide Time: 11:59)



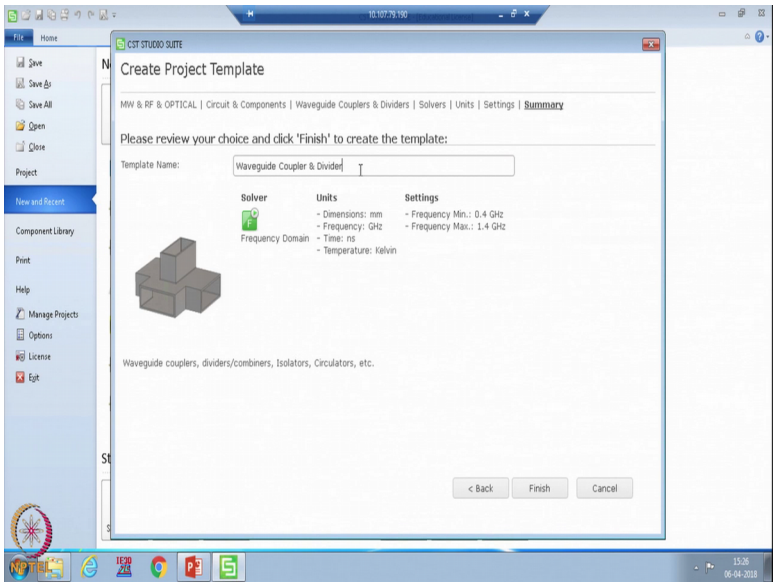
Here all the dimensions are already selected. If you want to change, you can change. Since, this power divider is designed for 900 megahertz.

(Refer Slide Time: 12:14)



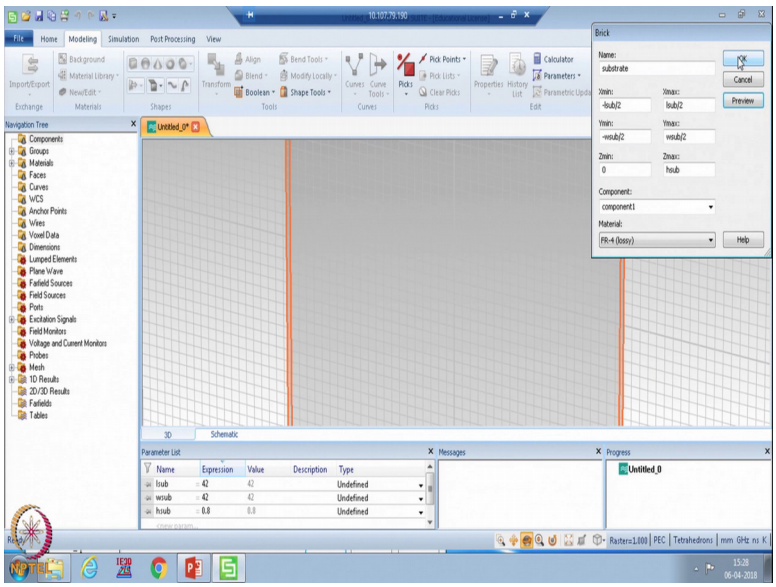
So, we will select the frequency range from 0.4 to 1.4 gigahertz, so that the center frequency will be around 0.9, and it will come in this particular range.

(Refer Slide Time: 12:27)



Then next, then you can give the template name, if you want. So this part, I will not repeat again, we will just simply go for the design part.

(Refer Slide Time: 12:38)

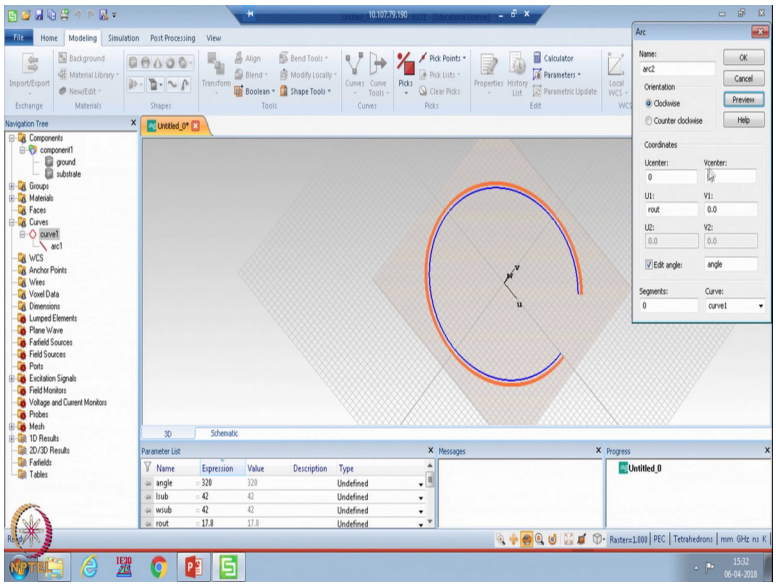


So, just to make this I as I said firstly, we will try to make the substrate first. So, to make substrate, go to brick, make substrate. Try to define all the parameters in terms of variables. So, here I will just give minus l sub by 2 for substrate length, I am using the parameter l sub. And for substrate width, I am using the parameter wsub. And as I told

the thickness is 0.8 mm. So, right now I am just defining all in terms of variable. The later stage, I will just give the values.

And, for to select the substrate, I have again used this material then from here select FR-4 substrate select a FR-4 lossy and then load it. Now, you give all the parameters. Maybe let us take substrate length as 42 mm, say width is 42 mm, and h of substrate as I told we will be using 0.8 mm.

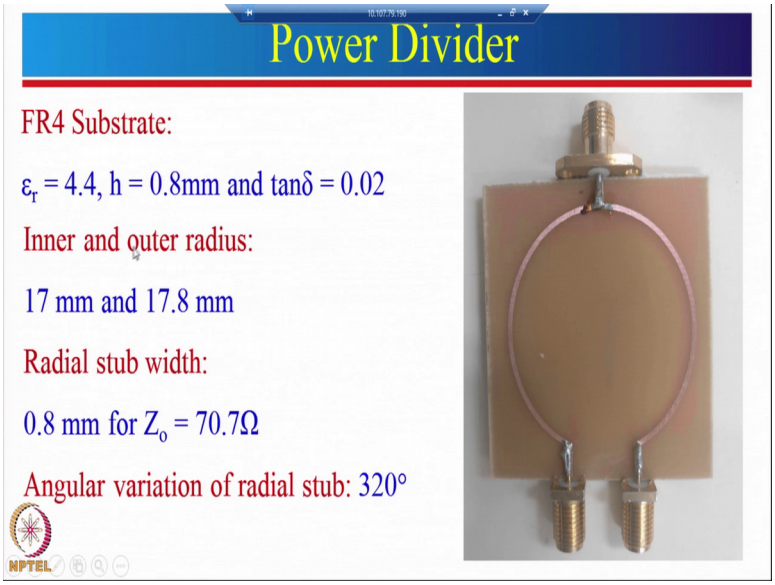
(Refer Slide Time: 13:56)



So, we have created a substrate. So, this is a substrate, which we made right now. Now, you can see all the parameters and corresponding details are already updated here. Now, suppose if you want to make ground, so you need to select one face. Just select one face, and then use extrude option to make ground, name it as ground give thickness as t. And by default in most of the fabrication lab the thickness of copper strip is 35 micron. So, we are taking as it 35 micron.

So, I did not change the material for this. For this copper the material should be PEC. So, I will change the material from here to PEC that is Perfect Electric Conductor, which will correspond to copper only. So, so far we have made the ground and the substrates. Now, we need to make the geometry of radial stubs. So, to do that again, you enable local coordinates system. Just select this face, and align WCS with it.

(Refer Slide Time: 15:14)



Now, we need to make the radial stubs. To make the radial stubs just to show you, here we will be using epsilon r equals to 4.4 and h equals to 0.8 mm. So, the inner radius and outer radius corresponding to 900 mm will be 17 mm and 17.8 mm respectively, so we will be using these radius. And the thickness of radial stub will be 0.8 mm, this is corresponding to 70.7 mm. So, you can use the impedance line calculator in order to calculate these width of the radial stub. And the angular variation, which we are taking for this is 320 degree.

So, just try to design this so, to design this go to curve use arc option. You can see here, this option is arc, select this option, then press escape. Now, see here just keep center as 0, and 0, and maybe give the parameter as angle. So, this let us take as 320, and for radius you need to select rin for the inner radius that maybe keep it as 0 give in as 17 mm. Now, you see it has created a arc circular arc.

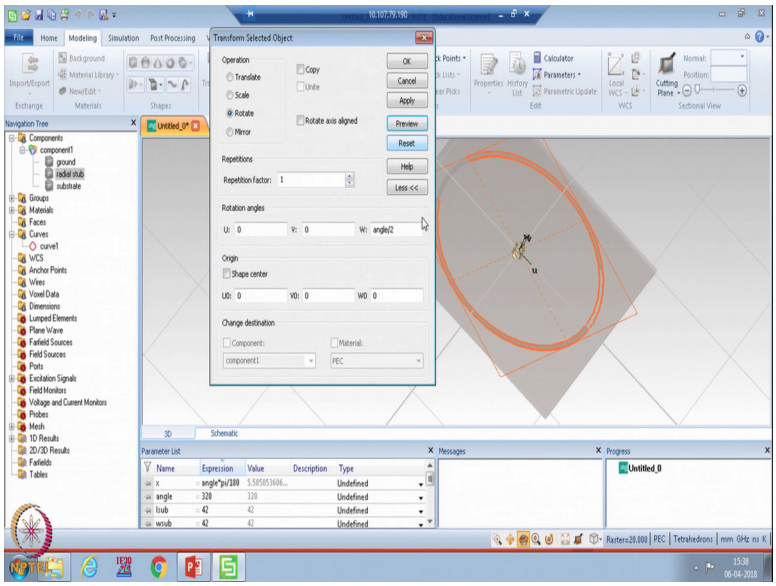
In the similar way create an outer arc use same option. Now, in this case use 17.8 mm instead of 17 mm, because this will correspond to the outer radius of the radial stub. So, give rout, angle you keep same. So, variable that we were using was angle, so I have just use the same variable. Now, rout just take as 17.8 mm. If you see here, it has created another arc of 320 degree angle. It is starting from 0 and going up to 320 degree. So, this is what it is then press . So, now you see, it has created two arcs of 320 degree angular variation. Now, you need to connect these two using the line.

So, just to make that just use this option line, again escape. So, one will be here. So, for that you need to use rin if you see in terms of coordinate, so rin this will be cos0 and rin sin 0. Similarly, for this outer part, it will be rout. Just use simple coordinate geometry to locate these points. So, I this should be 0 I have, this is rin cos 0, and sin 0, cos 0, and sin 0. Now, click on preview. So, you see here, it is creating a line over here, so it is connecting at this end. So, this is one line for another line, this coordinate will be rin cos 320. And this will be rout cos 320 and rout sin 320, if we see x and y coordinate system. So, we need to make the line accordingly.

So, again click on line, then escape and then just enter those enter those coordinates. So, rin cos maybe again I will give this parameter as x, then rin into sin x, rout cos x, rout sin x. Now, just define x here this angle inside cos, it take it as radians. So, we defined earlier it terms of degree just to convert it in radian. All we need to do is just write here, angle into pi by 180. So, now we have converted it in radian. Now, if you try to locate it, so here this is rin yes. So, you need to give negative of that this has this has created the line, now if you see here. Then press ok, now you see it has created the geometry.

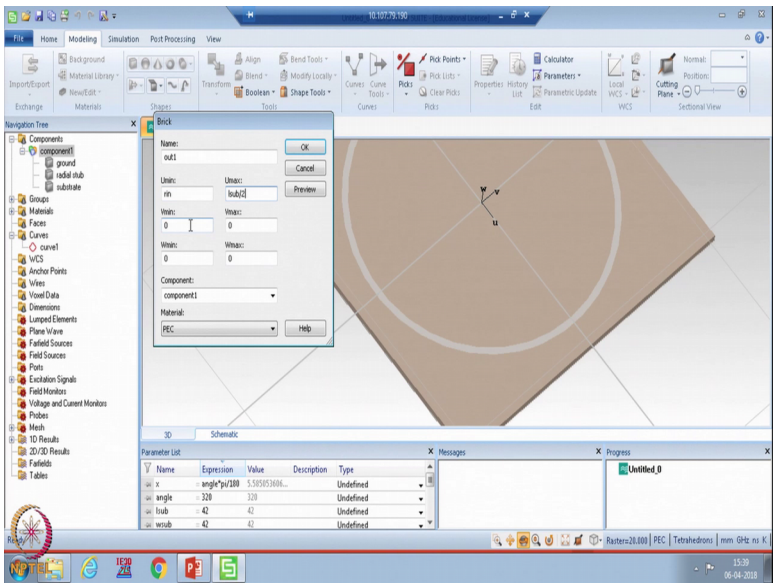
Now, this is just a arc. So, it has created the lines. Now, you want to make the polygon. To make the polygon, you need to select this full curve from here, just select the option curve and then use curve extrude curve and then double click there. Now, if you double click, it will create a polygon and it will ask for the thickness, which you want to make. So, just name it as maybe radial stub. So, thickness will be again copper thickness, so that will be t. And the material for this will be again copper that is PEC. So, now if you see here, it has created a radial stub. Now we want to align, so all we need to do is we need to just rotate it. So, just select this and use transform.

(Refer Slide Time: 21:53)



Then select rotate, and then in W you give angle by 2. So, now it has rotated, then press ok.

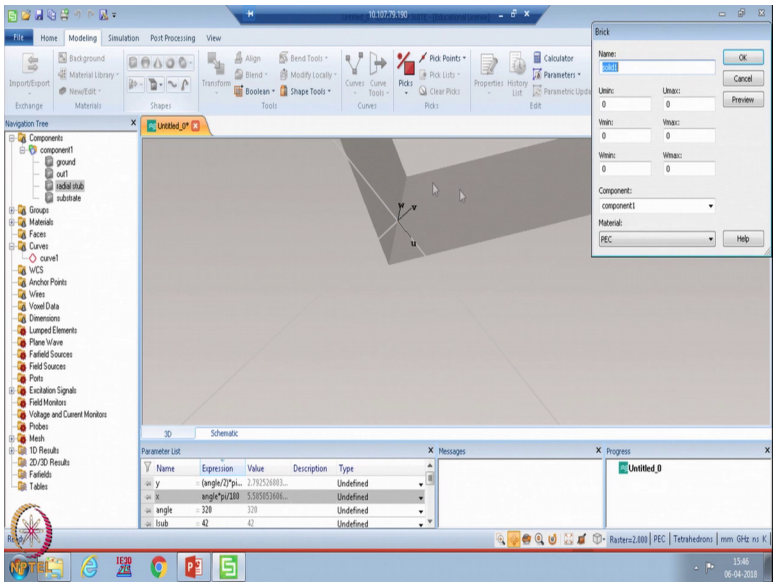
(Refer Slide Time: 22:12)



Now, if you see, it is aligned with the coordinate system. Now, we need to make the ports. So, to make port again select brick, define ports, so maybe just take as output 1. And if you see now, you try to make it from here. So, just make a strip from this end to this end. So, this is rin, and outer will be l sub by 2 ok. And for V this should be 50 ohm strip. So, for FR-4 substrate with 0.8 thickness, the w is 1.5 mm. So, for the time being

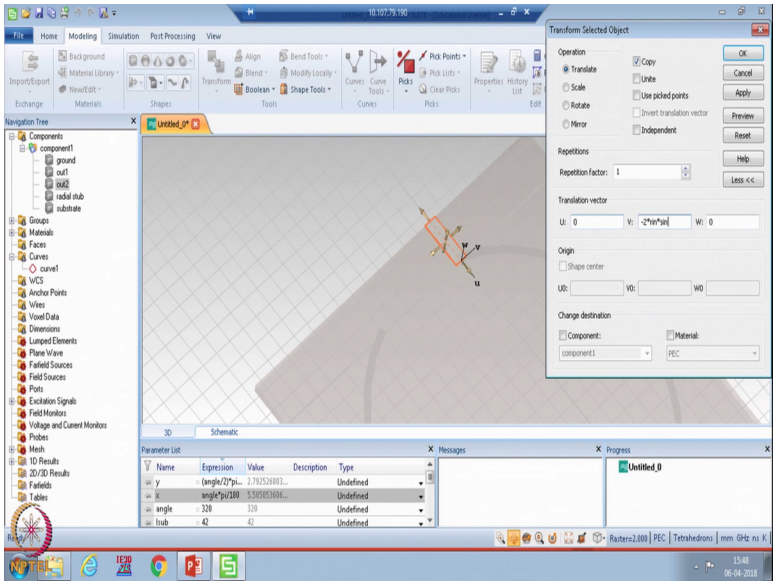
we name the variable as w 50. And thickness should be again t. So, w50 is 1.55, you can see this has created 1 microstrip of 50 ohm impedance line. So, in the similar way, you need to create the port at this end, select this edge. And then, you align WCS yes.

(Refer Slide Time: 23:39)



Now, you create the port name it as out 2, and from 0 to all you need to give is Lsub by 2 minus rin into cos y. And W it will be w50 by 2 yes. So, this is now port 2.

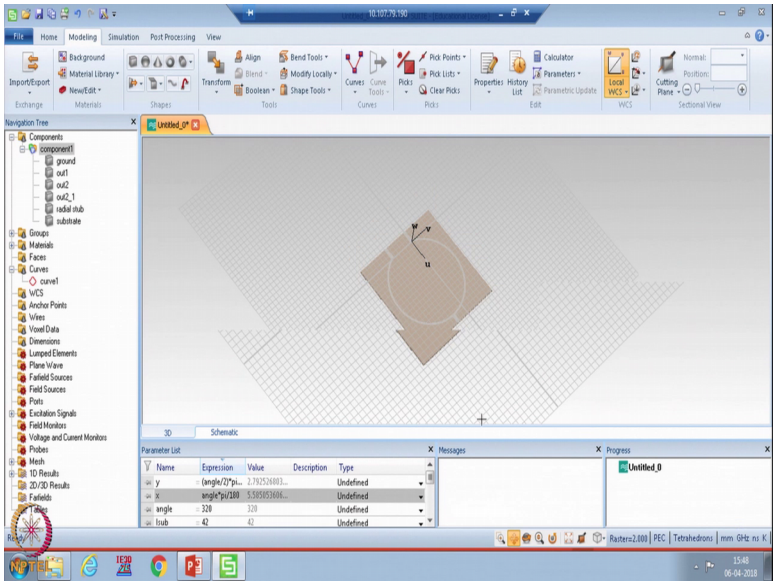
(Refer Slide Time: 24:03)



And for creating thickness, since we selected center. So, we need to take minus t by 2 and t by 2, so this is port 2. In the similar way, you can transform this port. So, just

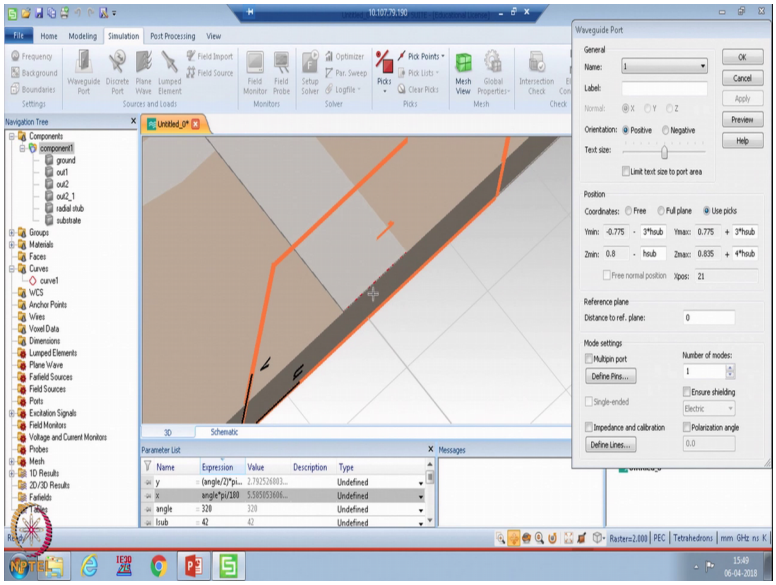
transform, copy, transform, you need to transform in y. So, in y if you transform, it will be minus 2 into rin into sin y, so this is port 2.

(Refer Slide Time: 24:59)



So, now we have created port 2. Now, you need to just locate the port, and do the simulation. So, just to locate it, just select this, select this edge, go to simulation, define port.

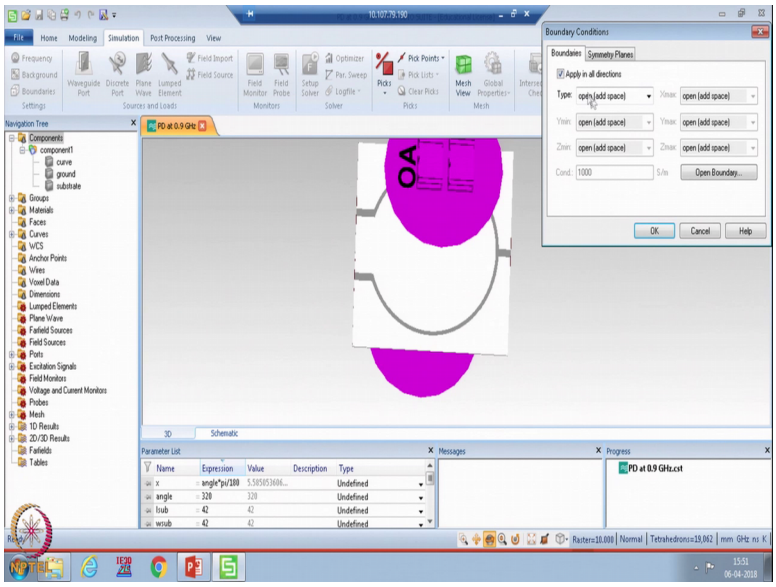
(Refer Slide Time: 25:13)



And as I told in the last class in the same way, you define the port at all the output ports. So, this is how you will create the port. So, in the similar way, you create the port for

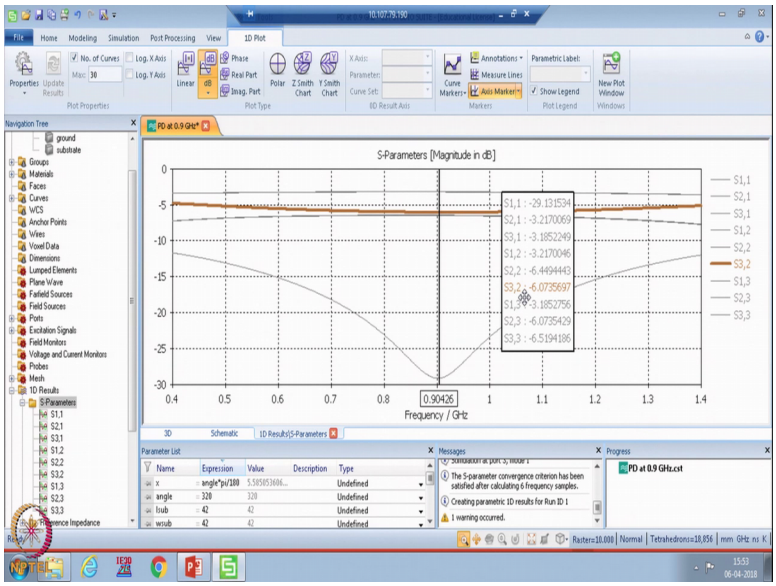
other two outputs. So, in that way, you will make the geometry, where you will generate all the output ports. In this way, you will create this geometry, you see here. This is output port 1, this is output 2, and this is output 3.

(Refer Slide Time: 25:41)



So, you have made the port you have made the geometry, next you need to do the simulation. So, to do the simulation go to simulation, again go to frequency setup solver. Before this, you need to give the settings, here you give the frequency range, if you have not defined it earlier, give it from 0.4 to 1.4, and then boundaries. Use open add space in all the planes, and then simulate it using frequency domain solver. Now, just wait for its response.

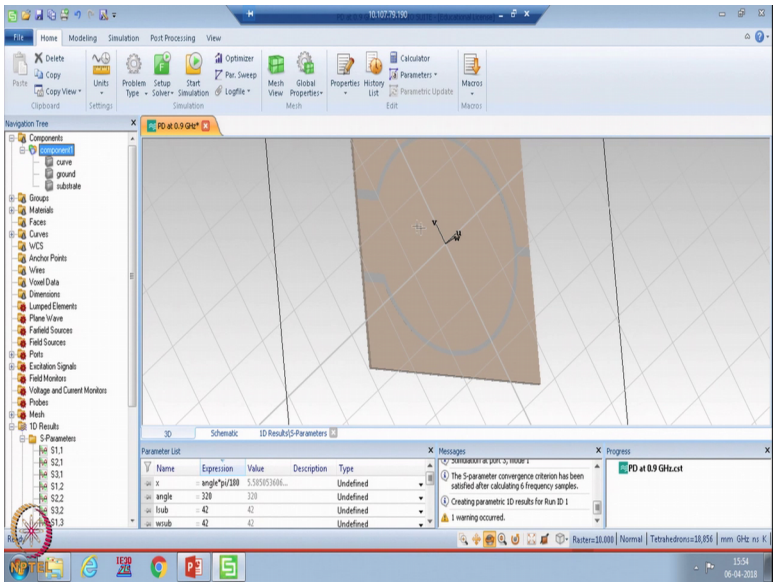
(Refer Slide Time: 26:28)



So, now it has been simulated, just to check the response, here you can see. So, this is S1, 1 this red one. So, you can see it is below 10 dB in all the frequency range from 0.4 to 1.4 gigahertz, and at 0.9 gigahertz it is resonating. Now, if we see S2, 1, and just try to locate the marker using this option, axis marker. Now, if you read the value of S2, 1, it is minus 3.21. So, ideally if there were no losses in the substrate, it should be minus 3 dB, but since the substrate is lossy.

So, it is giving is minus 3.21 dB output power. Similarly, at port 3 again it is giving 3.18 dB. And now I just want to highlight here. In the similar way all other results are coming, if you see yes. So, you can read all the values by just locating this marker. Just move this marker from here to here, you can read the value at the frequency point of your interest. Now, just I want to highlight here one thing, you see S3, 2 that is this now read the value of S2, 3 or S3, 2, it is minus 6 dB.

(Refer Slide Time: 27:54)



So, if you are giving the power at port 2, so the power that is going to port 3 is 6 dB. So, the isolation between these two ports is very poor. So, this is the drawback of power divider. So, if we give input power at these two ports port 2 and port 3, then in that way we will not get this sum at input port. So, to get this sum these two ports should be isolated.

So, in the next lecture, we will see how to design this power combiner. So, if we give power at these two port, how we will get the sum at this output port. So, in this particular lecture, we tried to design a power divider two way power divider. And we saw, how to design this power divider using radial stub. We use the different geometries using arc method, and then we simulated this power divider. And then we saw, how the power is divided in equal half at the output ports. So, in the next lecture, we will extend this geometry. And we will try to use this geometry as a power combiner by doing some modifications.

Thank you very much, we will see you in the next class.