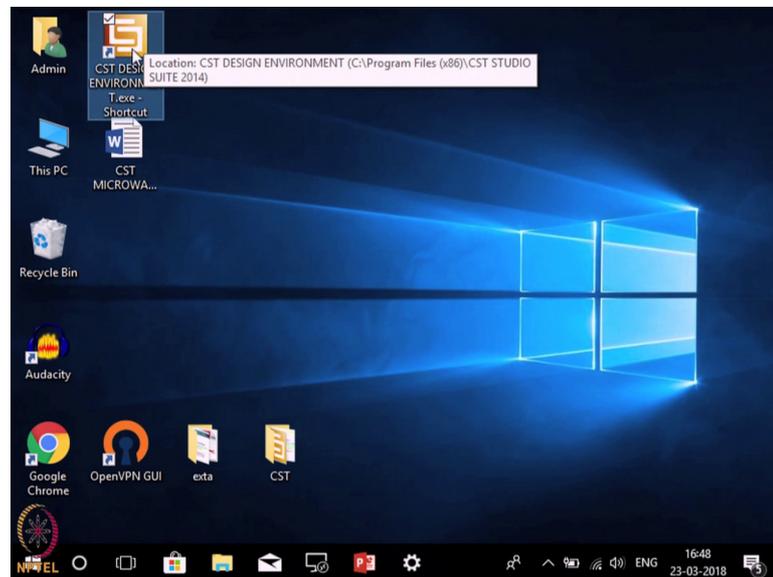


**Microwave Theory and Techniques**  
**Prof. Girish Kumar**  
**Electrical Engineering Department**  
**Indian Institute of Technology, Bombay**

**Module - 11**  
**Lecture - 55**  
**CST Software Introduction with Filter Design**

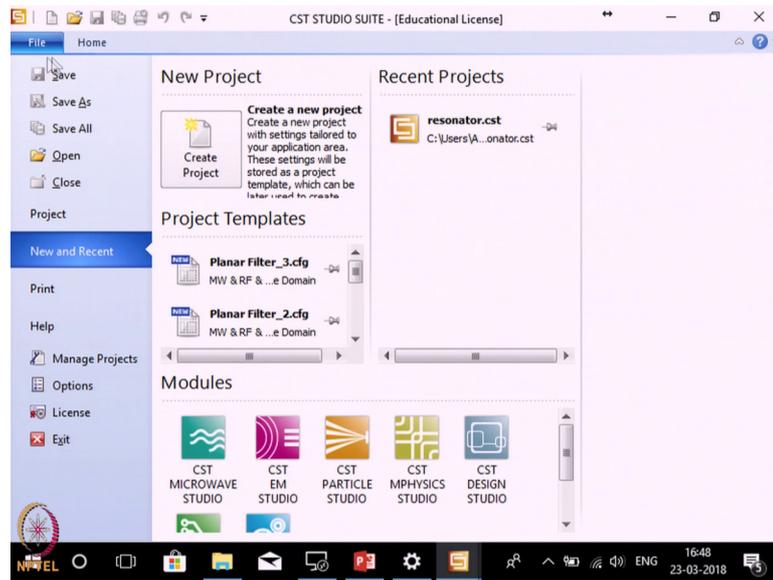
Hello today I will be taking a session on simulation software that is CST microwave studio, where CST stands for Computer Simulation Technology. Basically this software is a modeling package for high frequency simulations. So, let us starts CST software.

(Refer Slide Time: 00:43)



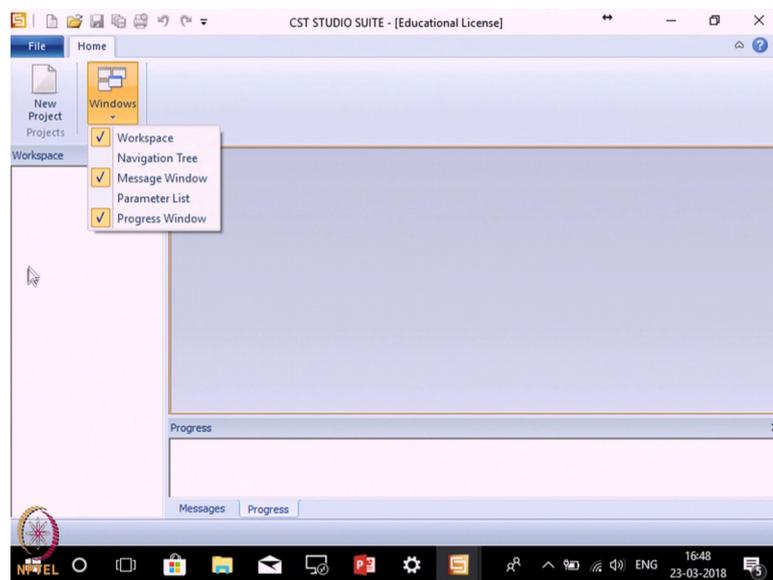
So, to do that open CST microwave studio so, when you open this you will see two menus, one is file other one is home.

(Refer Slide Time: 00:45)



So, file is see conventional menu, where you can save the file with your selected name or you can open the previously made files or you can create a new project and here you can see other information related to licenses and projects.

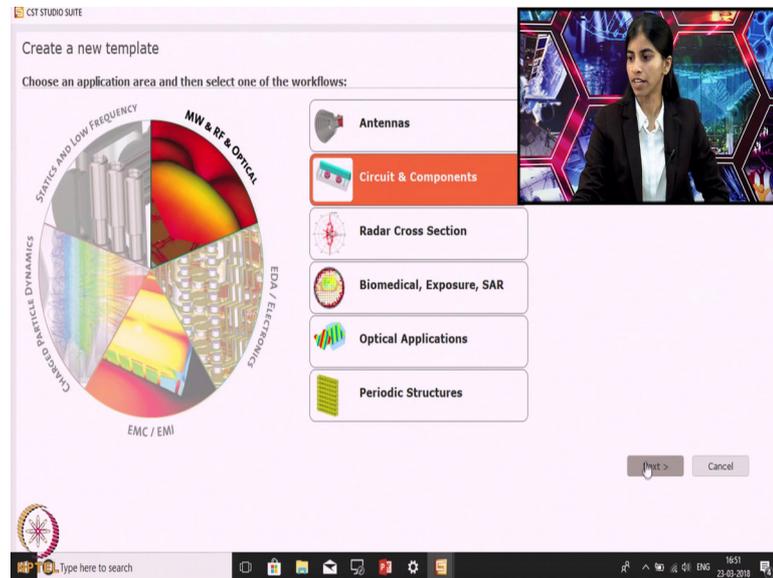
(Refer Slide Time: 01:10)



The another option is home, in this option you can create new project. You can see various windows; the windows which are shown over here is workspace message window and progress window. So, whatever is ticked the over here those windows are displayed here in this course. Firstly, we will tell about the software interface then we

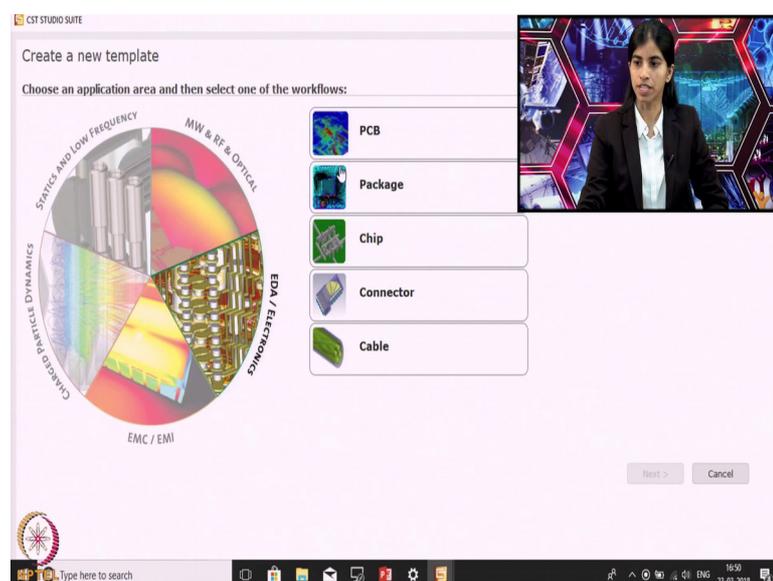
will design a filter at 1 giga hertz on a sub state FR 4 whose dielectric constant is 4.4 and thickness is 0.8 m m. So firstly, just create a new project to do that o click on new and recent project, then click on create project.

(Refer Slide Time: 01:51)



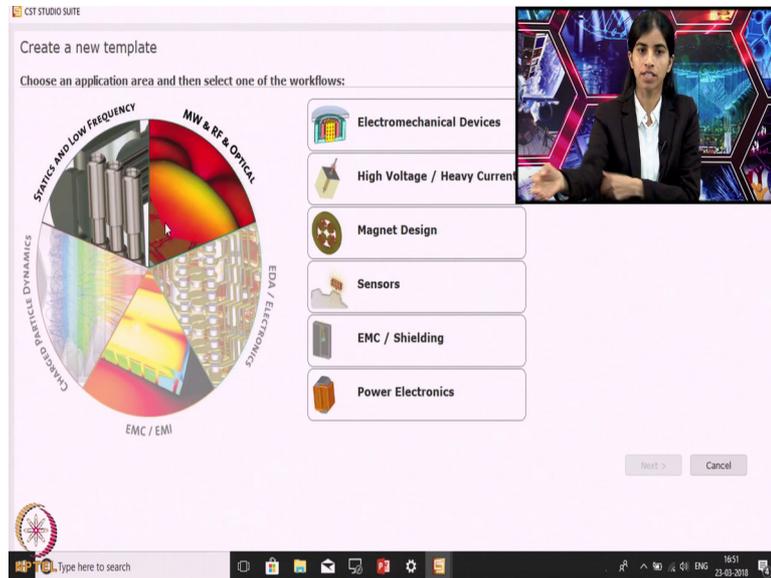
So, when you will create project you will see this window, where you will find various sub modules. So, corresponding to this sub module you can see various options you can design these type of circuits.

(Refer Slide Time: 02:04)



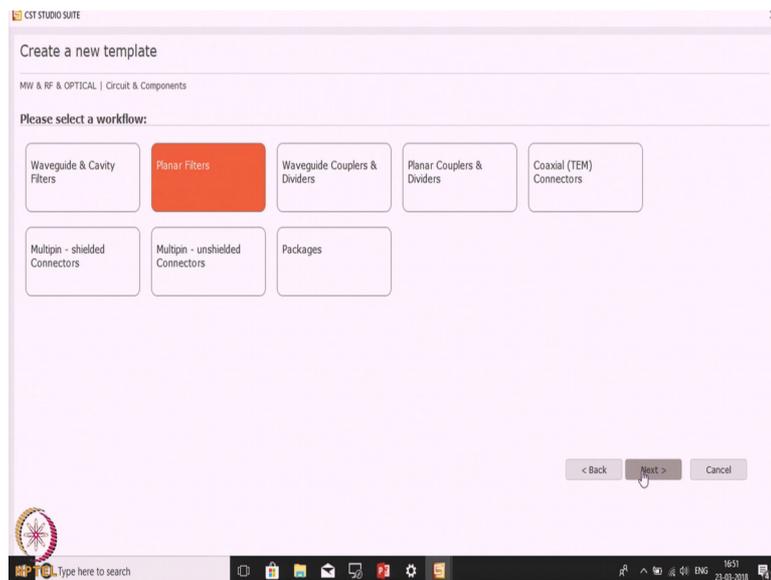
If you select this menu you can design PCB packages and other things similarly you can select various modules as per your requirement.

(Refer Slide Time: 02:10)



So, in this particular session we will be designing filter. So, we will be selecting microwave RF and optical modules and as we are designing filter. So, we need to select circuit and components, then click on next.

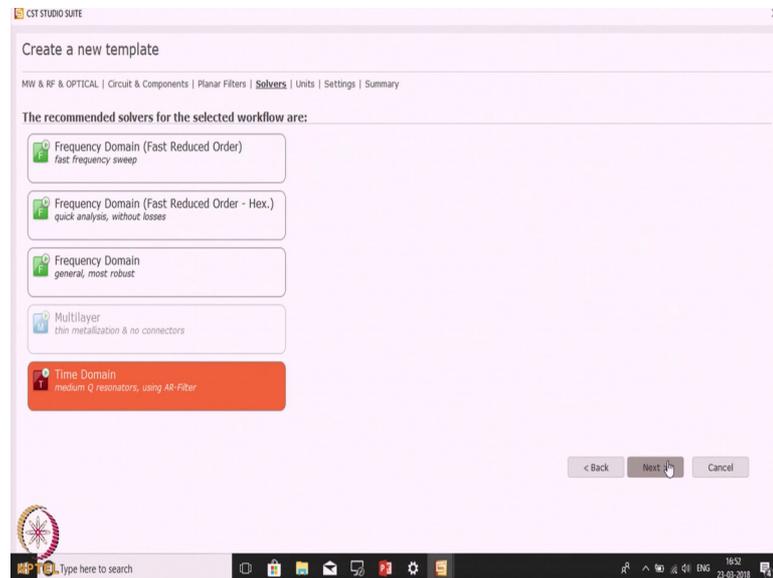
(Refer Slide Time: 02:26)



In the next window we will see various sub modules. Now I have already mentioned that we will be designing filters. So, we will be selecting this filter planar filter and if we

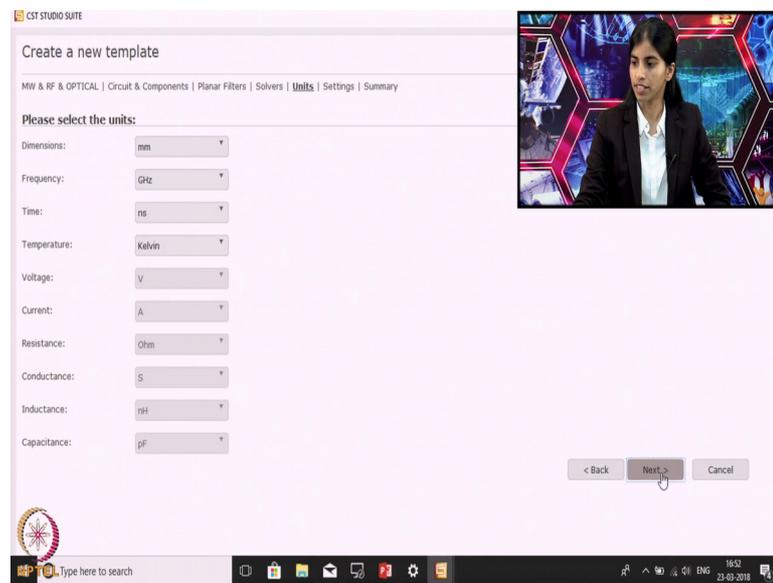
want to design waveguide, cavity, couplers and other things then accordingly we will select the relevant module.

(Refer Slide Time: 02:50)



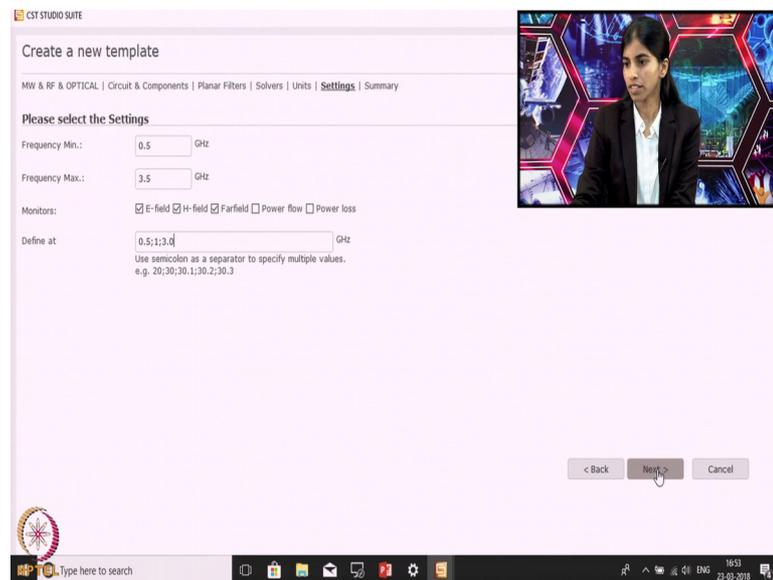
So, click on planar filter then go to the next option now you will see various solvers which it can use for its simulations. So, there are frequency domain solvers, time domain solvers and other then that few solvers are also there. So, this time domain solver is used for broad frequency range where as the frequency domain solver is more suitable for less frequency range. So, here we want to see the simulation for broad frequency range therefore, I would select time domain solver. So, click on time domain solver then go to next.

(Refer Slide Time: 03:23)



Then here we need to give preliminary settings like dimensions, frequency, time, temperature etcetera since I mention that we will be designing filter at 1 gigahertz. So, I will keep these dimensions as it is I would not change anything and then click on the next.

(Refer Slide Time: 03:43)

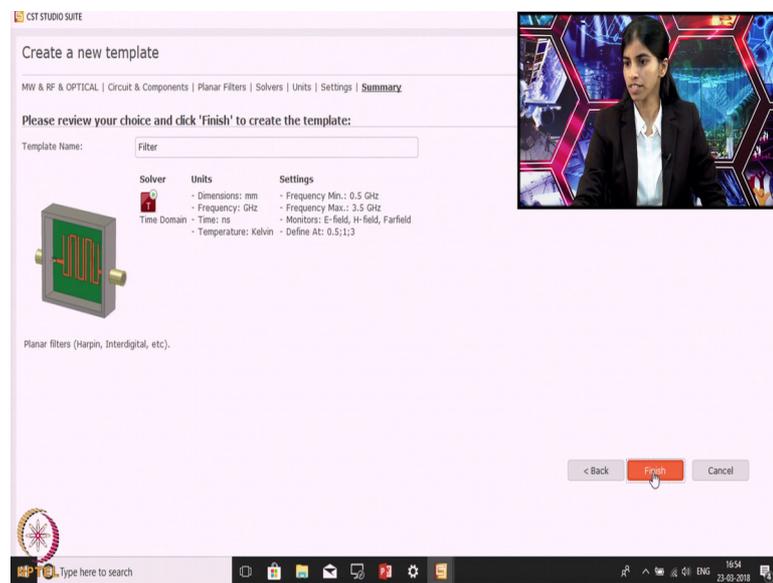


In the next section you need to define the frequency range for which you want to do the simulation. So, here I will give the frequency from 0.5 gigahertz to 3.5 gigahertz. Then if you want to see the E-field H-field or far field radiation pattern in between then you can

define your monitors accordingly. So, by default it takes the end frequencies and the centre frequency. So, when I clicked on E-field H-field and far field you can see here the three monitors are defined which are corresponding to the end frequency and the centre frequency.

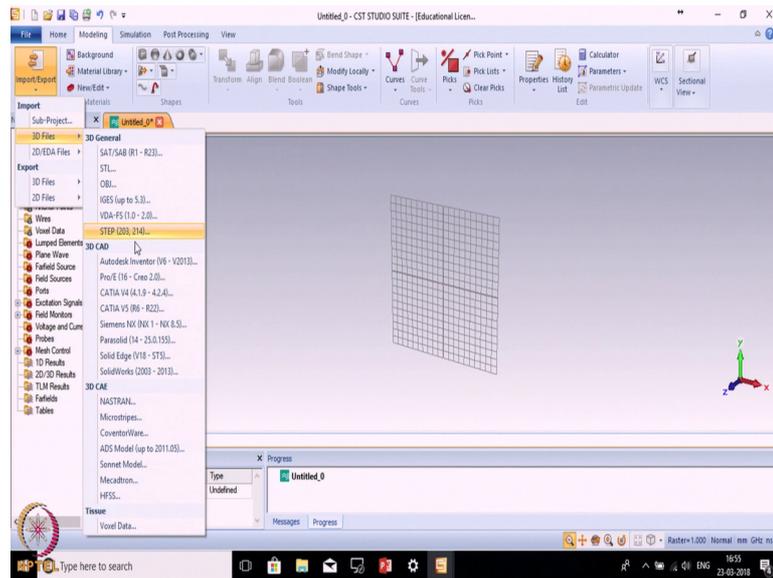
Now, since I said that I am designing filter at 1 gigahertz. So, I will keep one monitor at 1 gigahertz and other monitor I will keep on the third harmonic that is 3 th gigahertz. So, then click next.

(Refer Slide Time: 04:40)



Now, it has created a template with some default name. So, here you can rename as per your convenience. So, right now I will just rename it as filter you can see whatever template we have created the relevant information is given here. So, the dimension is in m m frequency is in gigahertz, time is in nanoseconds and the temperature is in Kelvin where as the frequency ranges from 0.5 to 3.5 gigahertz and the monitors at 0.5 gigahertz one gigahertz and 3 gigahertz. So, we have defined the preliminary settings, now finish this.

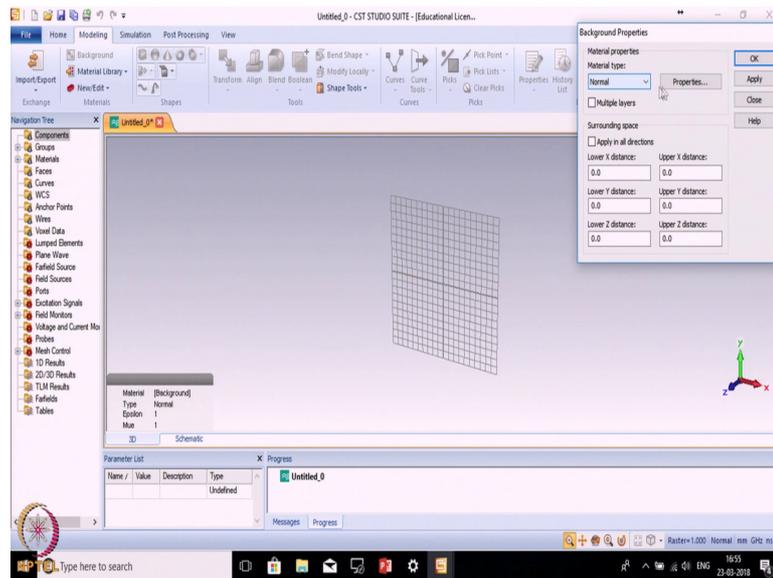
(Refer Slide Time: 05:17)



When you finish this so, it will create a temporary project file, you can see here various menus are there. So, one is file menu we have already explained about the file menu the next one is modeling menu, in this the first option is import. So, using this option you can import sub projects which were created in CST, you can also import various three d models like sat model IGS model etcetera whatever options are given you can easily import these models to create the geometry. If you have made this geometry in any other software similarly you can import 2D models in the given format.

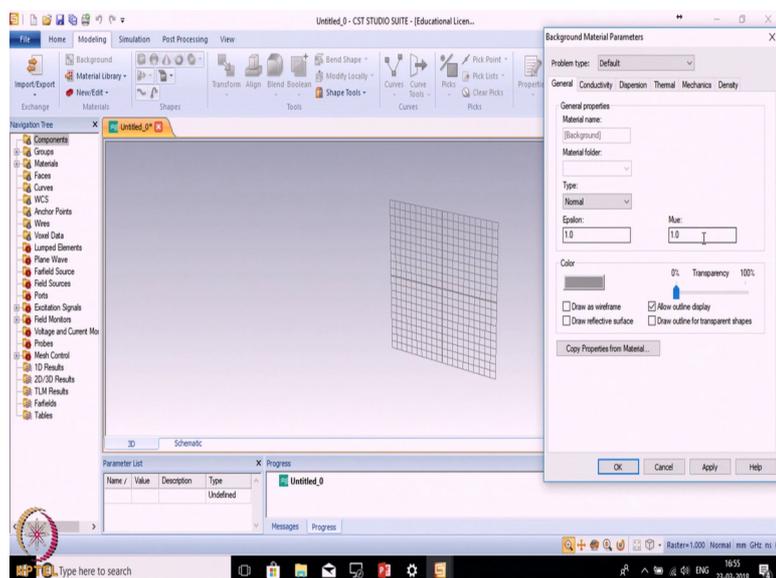
Similarly if you want to export the model from CST you can export in set format IGS format stub format and various models whichever is given here. And for today you can create Gerber file GDS file etcetera next part is background. So, background contains the background properties.

(Refer Slide Time: 06:16)



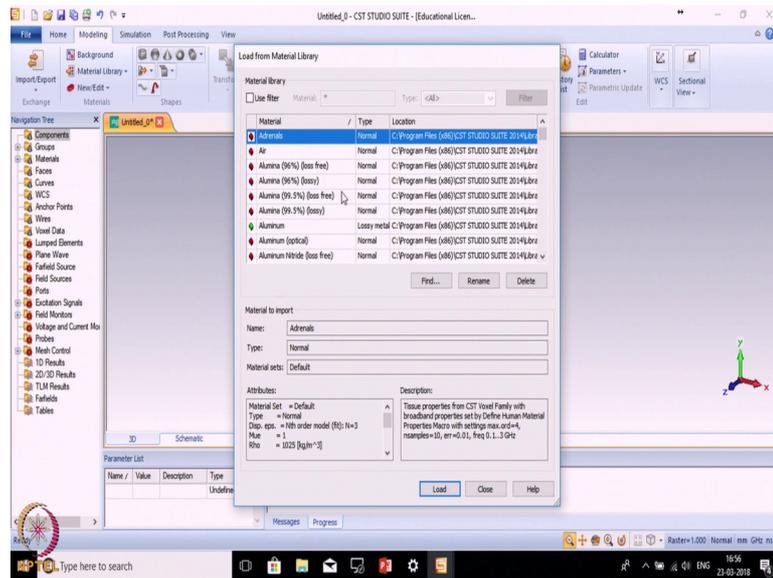
Here you can see by default background is normal. So, if you want to change the background material from properties by defining the conductivity epsilon r mu r you can change the material.

(Refer Slide Time: 06:24)



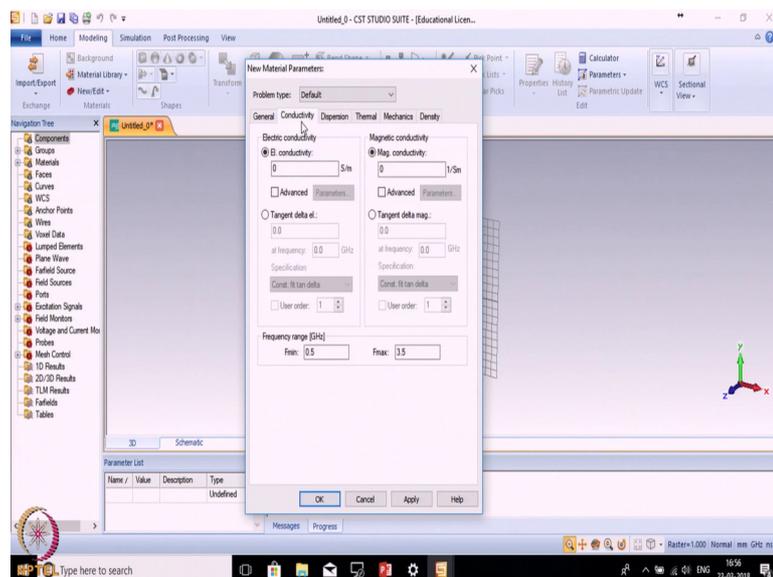
But here we want to keep the background material as air. So, we would not change anything next part is material library. So, by default CSD contains many materials.

(Refer Slide Time: 06:40)



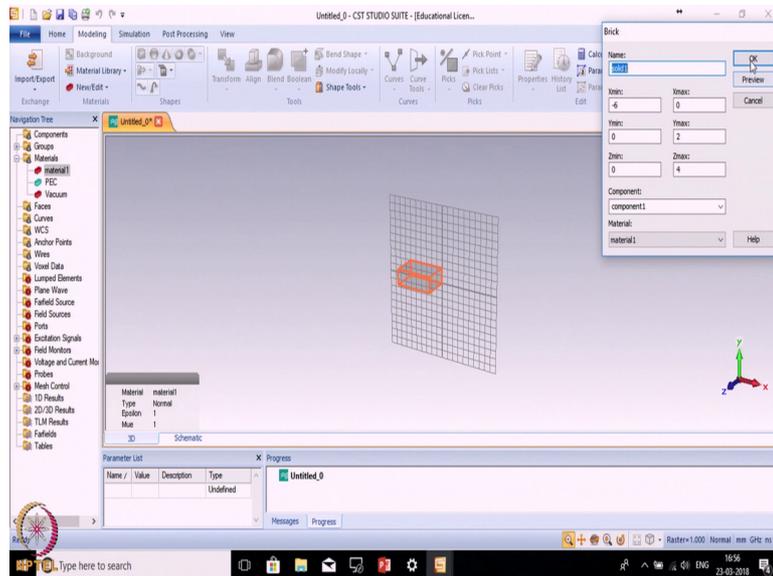
That are predefined and their libraries are imported in CSD. So, you can see the list of materials which are already defined in CSD, you can see here. Now suppose if you want to create a new library which is not given here you can make a new material library from here and then define epsilon r mu r conductivity and other things just to define the material library.

(Refer Slide Time: 07:01)



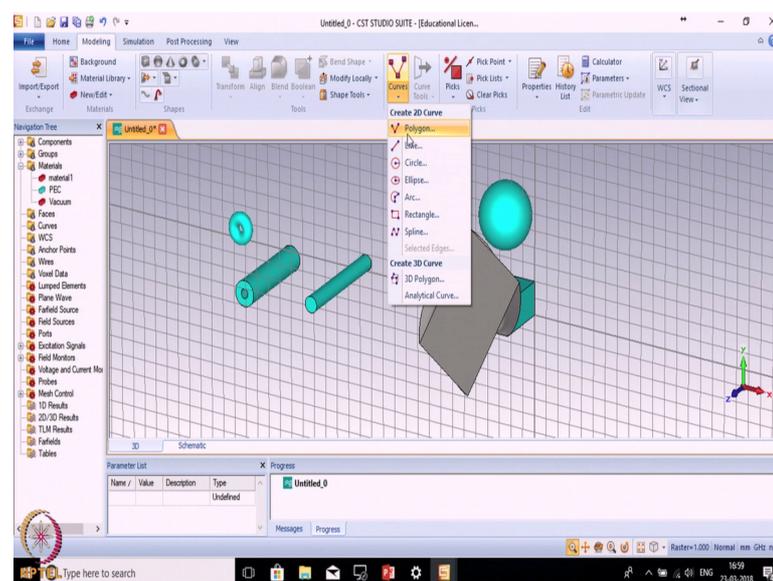
Then next option is related to the geometries here you can see various pre defined shapes are given. So, you can easily make these shapes just click on this, click on click here then you double click and make the geometry.

(Refer Slide Time: 07:21)



You can see wherever I click the mouse accordingly, it took the dimensions and it created the geometry corresponding to that. So, this is the geometry which I have created. So, this is a basic rectangular shape.

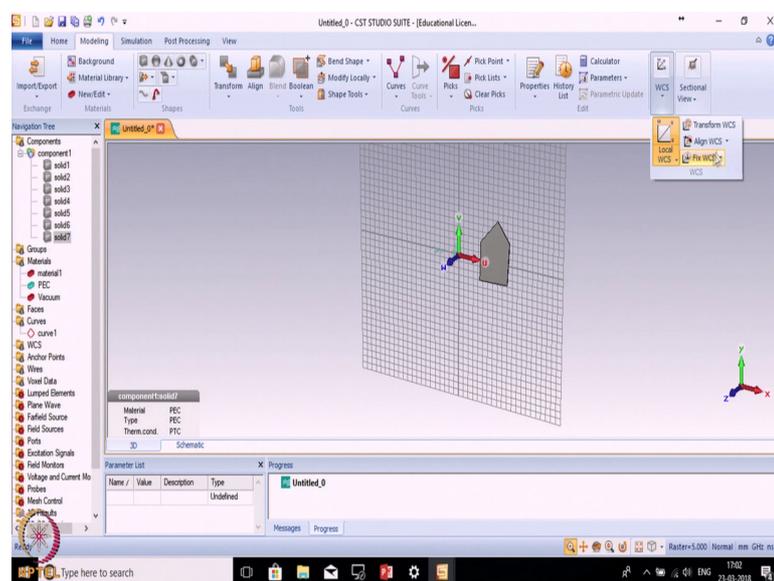
(Refer Slide Time: 07:36)



If you want to make this sphere you can also make this sphere, similarly if you want to make cone, you can make cones. Similarly if you want to make toroid, you can make toroid if you want to make a cylinder then you can make cylinder also just double click and then move your mouse in the height it will create cylinder as per your requirement. So, these are the predefined options, there are some other options like the extrude option to use extrude option you need to select a phase. So, to select the phase just move to the right side here one option pick is given.

In pick you can pick points, you can pick edges you can pick phases and other things whatever is given here. So, if I want to extrude it I need to select a phase. So, for example, if I select this phase and if I use extrude I can make a shape on this particular phase by giving some height let us say 0.5. Now I want this to be twisted let us say may be 60 degree and tapered by 70 degree and I can change the material if I want to change. So, suppose if I change it to PEC I can change it here then press you see it has twisted as well as it is tapered. So, by these options you can create various geometries. Suppose now if you are interested in some type of geometry which is not given here, for that you need to use this option curve like for example, you want to create a pentagon.

(Refer Slide Time: 09:42)



So, use this polygon option this, then click on next then next again next point double click and then double click. So, it has created a polygon, but it contains only wire you can see here these all geometries were 3D polygons. So, corresponding to these polygons

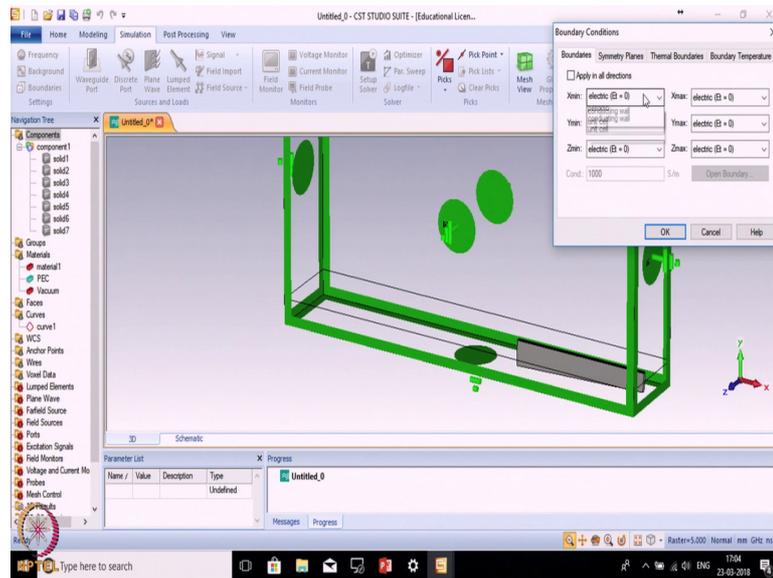
one solid is created over here you can see; however, this is wire. So, related to this details it is given in this curve, Now, if you want to make a polygon using this for this you need to select this curve tool use extrude option extrude curve, then double click on this and give a thickness for which you want to create a polygon.

Suppose I will give here 0.5 may be then it will create a polygon. Now you can see corresponding to this one polygon has created which is solid 7. So, in this way you can create a polygon of your desired shape. So, these are the options now if you want to select a edge, you need to select pick edge and then select the corresponding edge double click. So, in that way you will select the corresponding edge similarly if you want to select the middle point of a edge or the endpoints, you can use the relevant options from here.

Now, if you want to clear this pics use this option this is the property option this gives the information about the objects which were created in the workspace. So, this is the workspace where we create a full geometry. The next is history list this option is very useful when we want to debug a particular geometry if there are some errors, then this particular option is very useful to debug the errors which we have made. I will talk about this option little later the next is calculator here you can give expression and it will give you the results maybe suppose if I take let us say 20 into 30 then. So, then this product it will create and the result will be 600.

So, in the similar way you can give various expressions and it will give you the value corresponding to that. Then next is the WCS; WCS here by default in CSD there are two types of coordinate system, one is global co ordinate system another one is local coordinate system. So, this x y z is global co ordinate system, we cannot change this co ordinate system. So, if we want to change co ordinate system to a particular location, then we should use this local co ordinate system. So, in general I would suggest whenever you make your design you work using these local co ordinate system. Now if you want to align these local co ordinate system with edge or particular point you can do with these options easily. The next one is the cross sectional view.

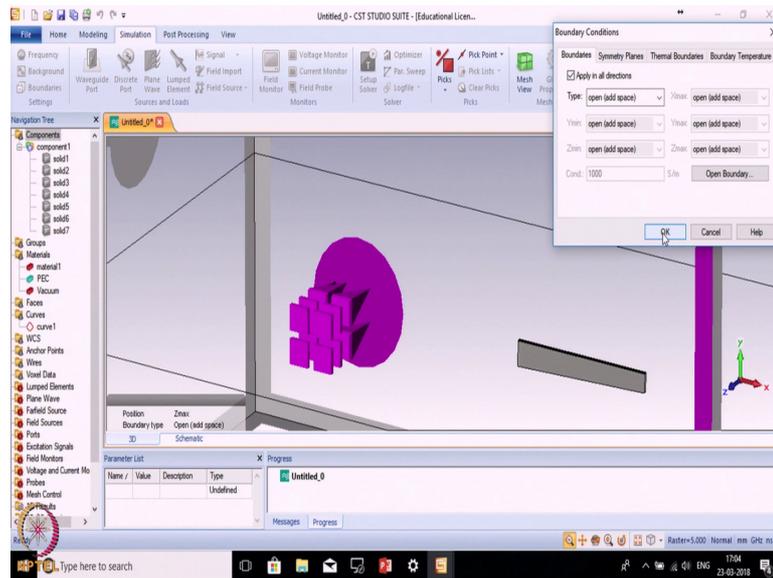
(Refer Slide Time: 12:59)



So, if you want to cut and if you want to see at particular instant, you can use this particular menu here in x plane you can see this it is shown using the cross sectional view of at x equals to minus 3 you can change the location similarly you can see it in y plane and in the same way where ever you want to see, you can see the next option is simulation. So, first one is frequency. So, if you have not defined the frequency earlier, here you should mention the frequency next option is background. So, in the background I have already told we will be using normal, you can change here if you want to change it at later stage.

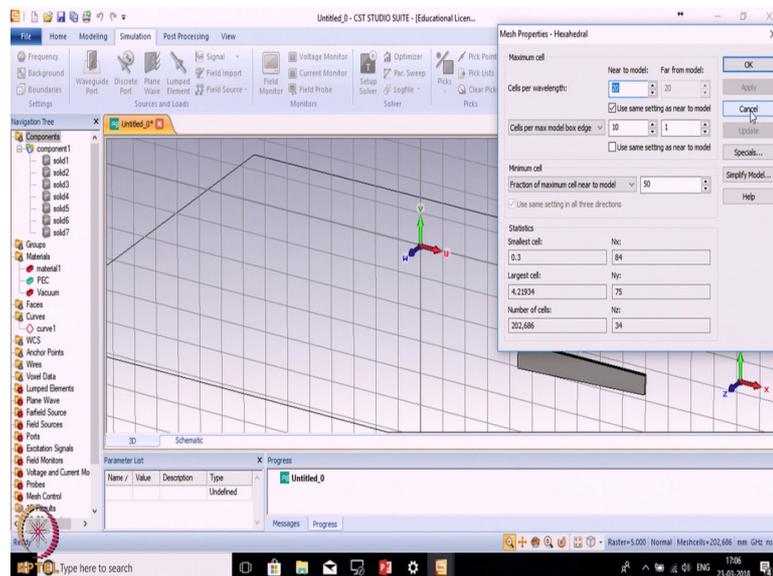
Next is boundaries so, in our case will be using boundaries. So, here you can use various type of boundaries like perfect electric wall, perfect magnetic wall open add space etcetera. So, we will be using open add space. So, I will make those changes in all the directions.

(Refer Slide Time: 13:57)



Then these information is related to port.

(Refer Slide Time: 14:01)



So, in case of micro strip we will be using the waveguide port and in case of wire antennas we will be using discrete port the next part is lumped elements. So, in lumped elements we can provide RLC components diodes we can provide the specifications as per our requirement and correspondingly, it will give you the simulated results.

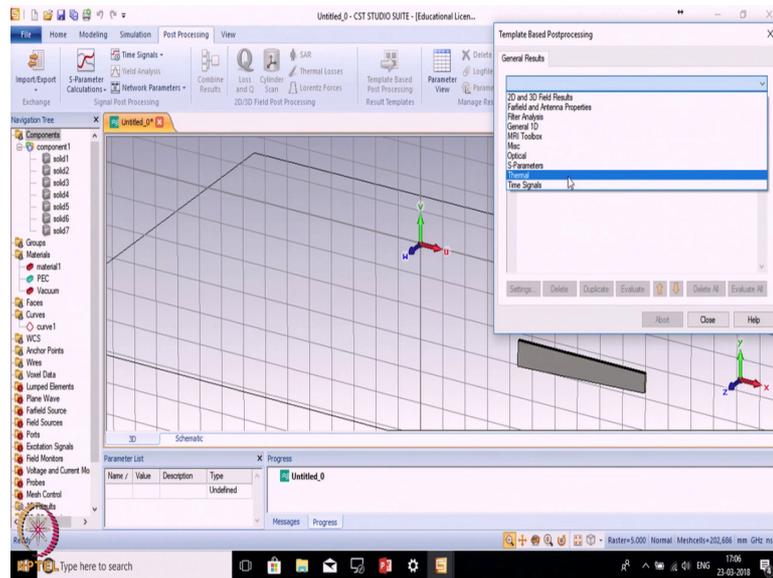
The next is field monitors so, if you want to define a field monitor for E - field H-field surface current or various options whichever is given here, you can define it for a

particular frequency point. The next is voltage monitor current monitor. So, you can apply all these monitors as per your requirement the next is the transient solver which you have selected, the next option is optimizer. So, by default it provides few optimization algorithm like you can see here CMA evaluation and this trust region framework. So, these algorithms are used to optimize if you want to optimize it for a particular function, you can define that function over here.

So, these options if you want to explore you can use it. The next section is parameter sweep, I will talk about this little later, the next is lock file which will show the results again I will talk about this later these options I have already explained this one is mesh view, which will show the meshed view of your structure the next is global properties. So, if you want finer mesh or maybe the mesh size of relatively more size, then you can change these settings as per your requirements, but for the time being I would not change these things next section is intersection check. So, if some polygons are overlapping with each other then this particular section intersection check should be used.

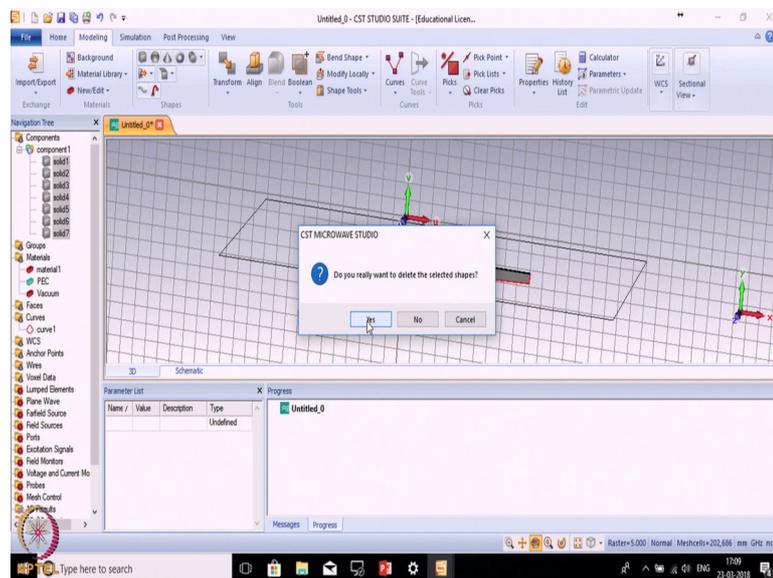
Another one is electrical connection. So, this particular option is used to check the electrical connectivity between the polygons. So, this particular window is for the post processing parameters like s parameters, gain far field and other things. So, these are the option you can try from even template based processing you can see you can select these options.

(Refer Slide Time: 16:16)



2D far field results far field antenna properties s parameters and all those things and accordingly you can do the processing.

(Refer Slide Time: 16:27)



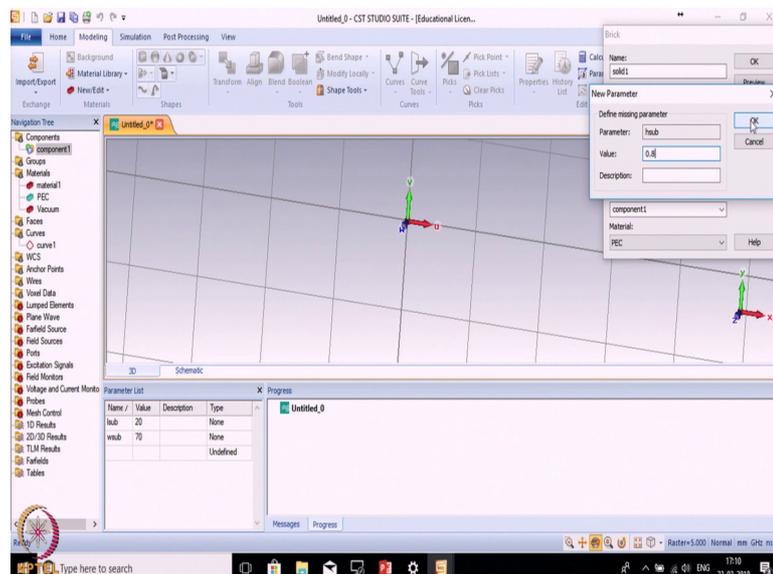
And last option is related to the structure view. You can view this the structural design in different ways by just rotating or by zoom in zoom out on other things you can see here. In this option here you can see in windows you can select the window of your interest whatever you want to see suppose if we I will and click this window, you can see I cannot see this navigation window and if I click it appears again. So, I just want to tell

you in this navigation window, these options to this point these are related to the modeling of the structure and below this these are related to the post processing parameters, which shows the results of the design. This one is a parameter list window here we define various parameters, here you can give the variable name and the values and we can define their type to remember at the later stage.

The next is a message window which displays the various types of errors or alerts or warnings whatever comes in the simulation. Next part is a progress window which shows the status of the simulation when we run it whether it is completed or not; let us start with the filter design. So, as I mentioned that we will be designing filter on FR 4 substrate, the resonance frequency of the filter let us take it as 1 giga hertz. Firstly, to define a substrate go to modeling and we know a substrate is basically a brick before this I would just suggest that whatever geometries we created just delete first all those geometries.

Because, those geometries were created to give you a reference that what type of geometries can be made using this particular interface. So, press parameter on brick then scrip then you can see here the local coordinate system is active.

(Refer Slide Time: 18:30)

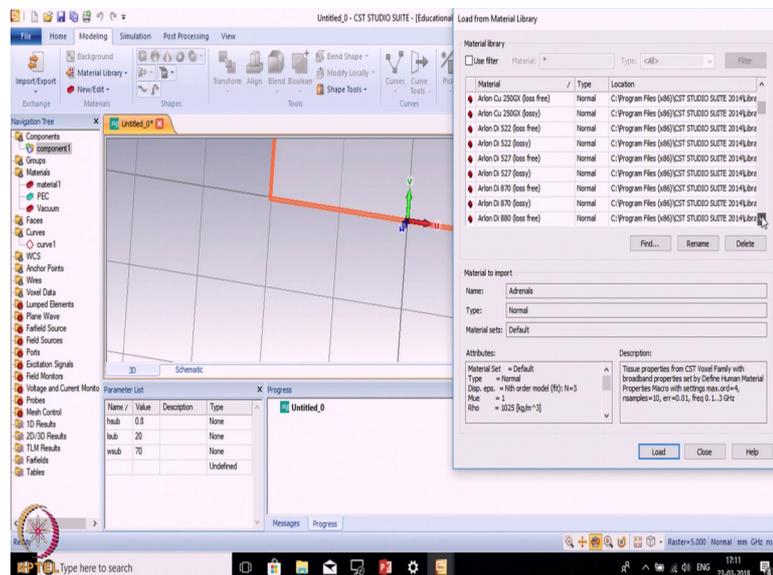


So, in u direction let us take it is a length and in V take it is as a width. So, for length now I would just suggest all of you to define these lengths and widths in terms of variable parameters. Because here if you define it in terms of constant then at later stage

if you want to change it would be more time consuming because at every stage you will have to change it manually while if you define the variable.

You need to just simply replace in the variable it will automatically change, I will show you at the later stage. So, let us just define it as a substrate may be l sub as variable name then the next is just keep this term coordinate system as symmetric and then define it from 0 to w substrate, and the thickness of the substrate let us take it H substrate. Now I will define these parameters l substrate I am taking may be 20 w 74 and then h we are using 0.8 m m thick substrate.

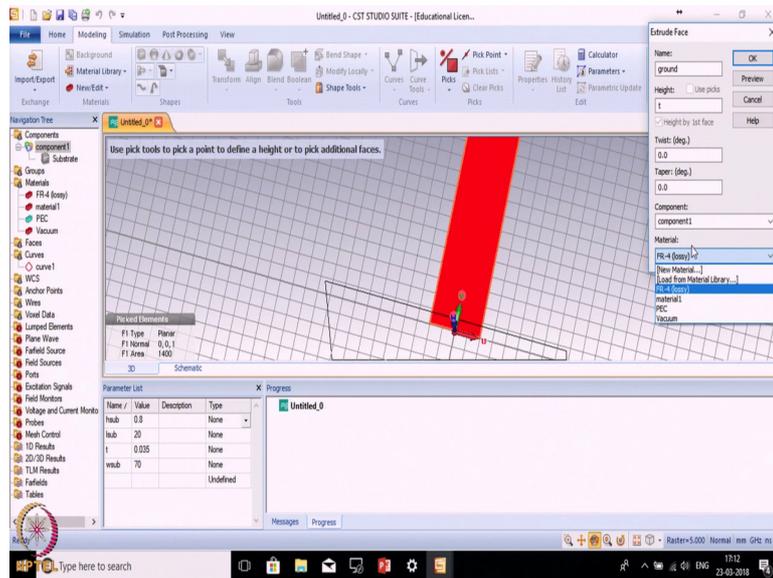
(Refer Slide Time: 19:52)



Now, here you can see the component is one, you can change to another component and if you want to keep it in the same component you can keep in the same component.

And I want to rename the solid as substrate, the material since I told you that we are using FR 4 substrate. So, we need to see whether we have FR 4 substrate in the library or not. So, we need to just check it in the library just scroll it down to check it ok. So, FR 4 lossy you see we have selected.

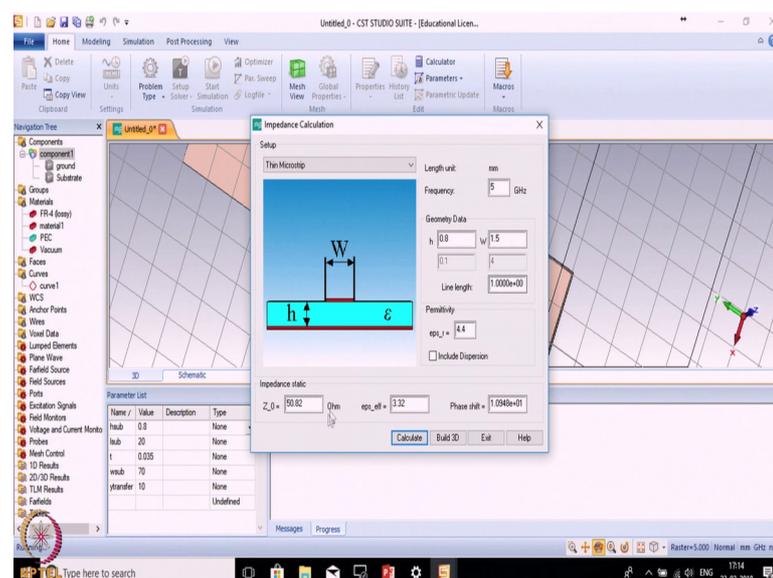
(Refer Slide Time: 20:37)



This is the substrate we have defined now we need to make a ground plane on bottom side. So, just select this phase and then use extrude option and name it is as ground and give a height thickness is t for the copper, s and use t as 35 micron.

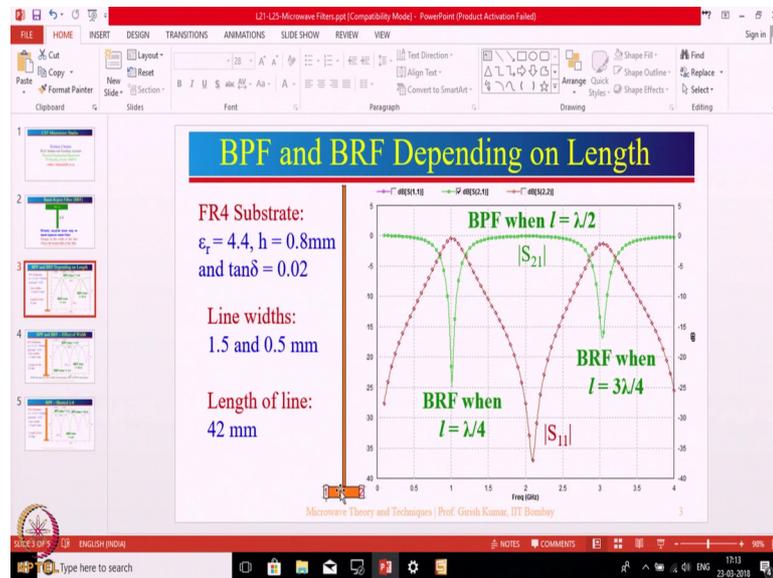
Because when we get it fabricated in general, 35 micron thick copper layer is printed and the substrate material should be perfect electric conductor for this case. So, we have defined the ground plane and the substrate now we need to define the micro strip.

(Refer Slide Time: 21:33)



So, to do that first start here and then align WCS, then we want to may be start from somewhere inside. So, for that you can use transform you can use y transfer and maybe just give this value as 10 just see this geometry here.

(Refer Slide Time: 21:51)

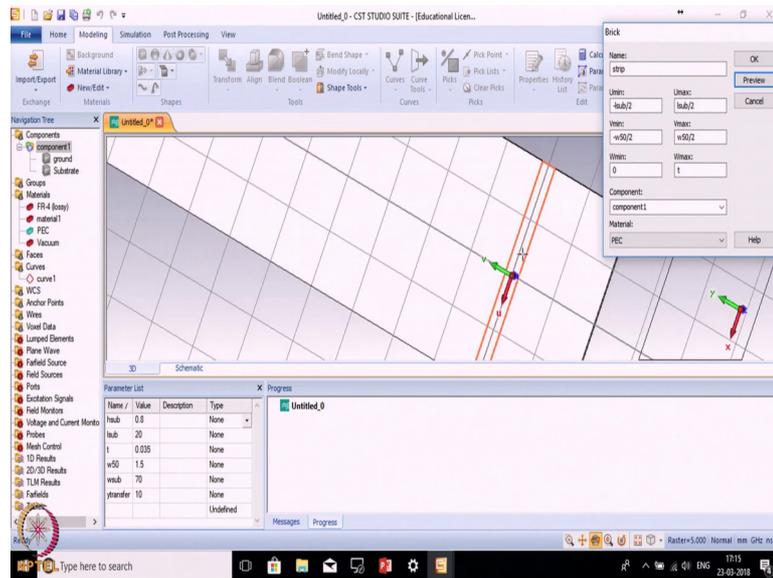


So, in this particular bottom strip is of 50 ohm impedance. So, we need to find how much will be the with corresponding to 50 ohm impedance.

However it is given here it is 1.5 ohm 5 mm, but I will show you how to calculate using CST. So, here if you check in home there is a option macros you go to calculate then use calculate analytical line impedance. Then go to thin micro strip option and here various options are given. So, you can calculate the various lengths and widths corresponding to various options like coaxial strip lines and other things to select thin micro strip line. Give a epsilon r 4.4 and the thickness we know it is 0.8. Now just check for w equals to 2, how much will be the impedance it is 42.48.

Now, we know w and z are inversely proportional. So, if we reduce the w my impedance will increase. So, let us take maybe 1.5. So, you see corresponding to 1.5 it is 50.82.

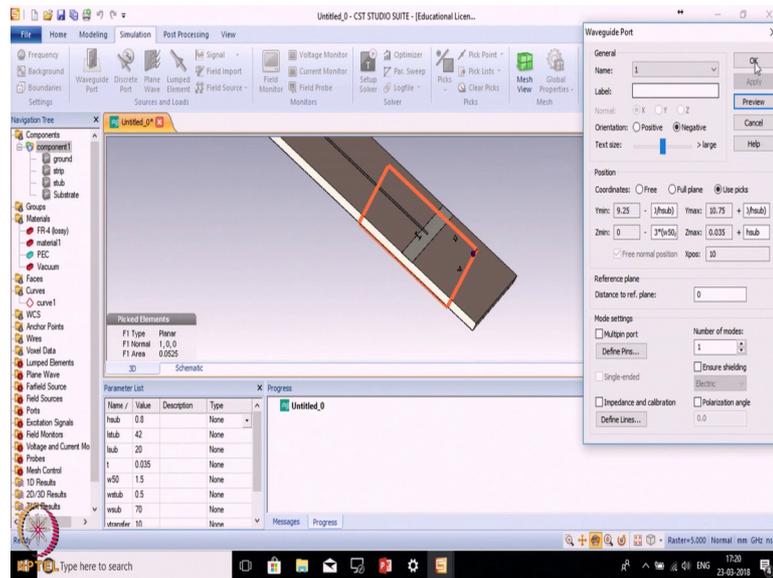
(Refer Slide Time: 23:04)



So, from here you can calculate the thickness of 50 ohm strip. Now we need to make 50 ohm strip. So, to do that just name it as strip minus maybe take same as  $l_{sub}$  substrate length  $l_{sub}$  by 2 and then minus  $w$ . I am just giving the variables and I will put the value of these variables. So, thickness of this copper you take as 35 micron again.

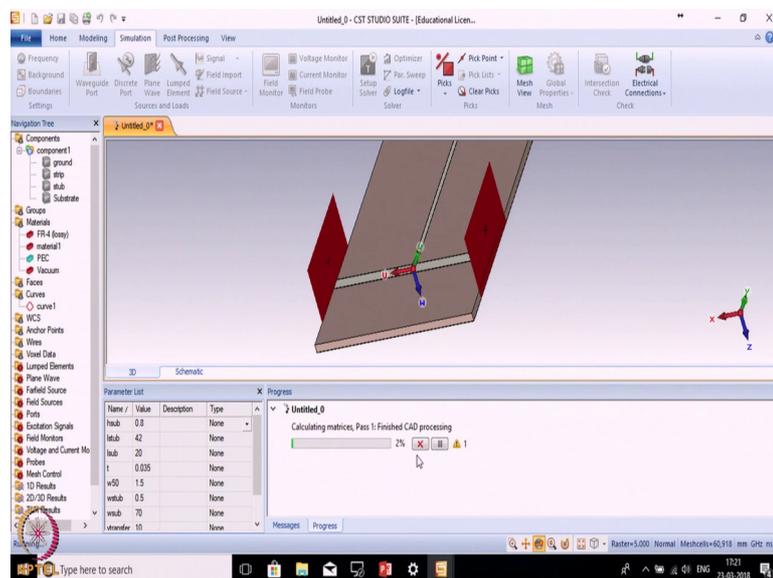
So,  $w_{50}$  will be 1.5. So, we have created this strip for 50 ohm line you see. Now we need to create the lambda by four stub. So, to create the same again use strip and just name it as a stub maybe from let us take the width of the stub as 0.5 mm. So,  $w_{stub}$  may be give variable name as  $w_{stub}$  by 2 from this. So, let us take the width of the stub as  $w_{stub}$  and accordingly define, and  $l_{stub}$  corresponding to lambda by 4 using that same micro option it will come around 42 mm. So, will define the same so, let us take it 0.5 and then  $l_{stub}$  of 42 then we should define port.

(Refer Slide Time: 25:01)



So, to define port we need to select this phase, then define port this should be 3 into ratio of 50 ohm strip width and h substrate yes; in the similar way you define another port which is this side yes and then you start the simulation we will take some time.

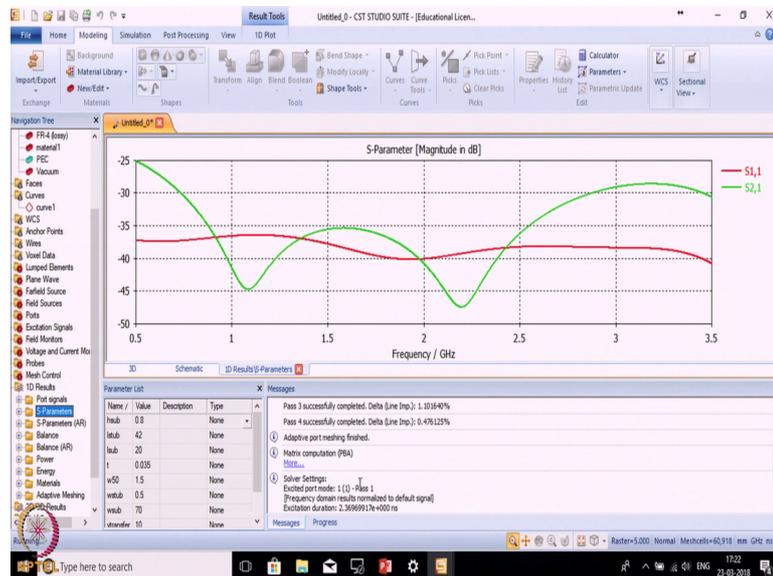
(Refer Slide Time: 25:39)



So, I just want to tell you that we have used the lambda by 4 strip line and it is open at this end. So, for the lambda by 4 strip line we know that it inverts its behavior. So, now, if it is open, it will act like a short for the frequency for which it is lambda by 4. So, all the currents will go to this end.

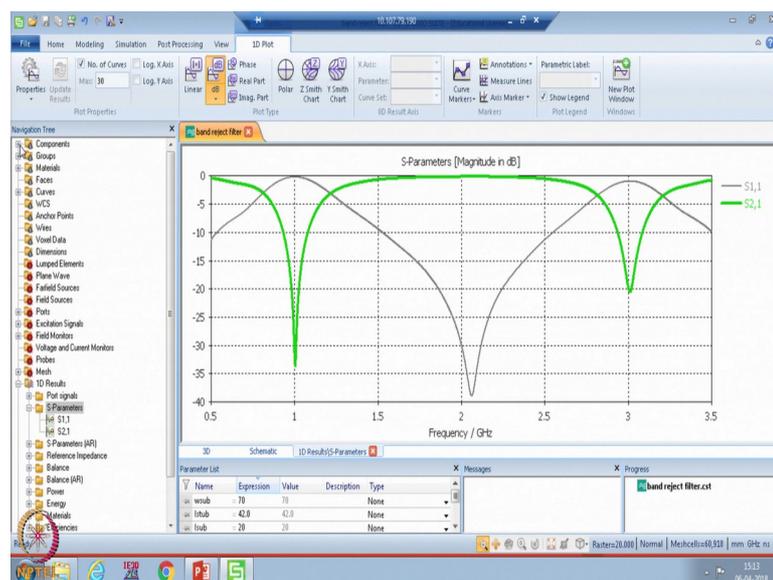
So, corresponding to that frequency, it will act like a band reject filter. So, the response should act like a band reject filter will see the response.

(Refer Slide Time: 26:24)



So, in between as I was mentioning, in the progress window we can see what is going on the progress in the message windows we can see various warnings and all those things, in between we can see the simulation results from the s parameter window you can see here, the simulation is still going on it will take some minute here.

(Refer Slide Time: 26:46)



So, we selected  $w$  stub 0.5. So, this will be the response corresponding to 0.5, you see stub 21 it is showing a notch which cause which means that it is a band reject filter and  $s_{11}$  is very high.

So, it shows that at 1 gigahertz it acts as a band reject filter and at third harmonic it will behave in the same way. And while as if you see at 2 gigahertz that is up to twice of this frequency the length will be  $\lambda$  by 2. So, open will act like a open so, it should act like a band pass filter. So, it is showing the same behavior as band pass filter should do.

So, this is how we saw that how this  $\lambda$  by 4 direct coupled resonator will act like a band pass filter or band reject filter. So, with this I would like to conclude; today we discussed about the interface of CST microwave studio, then we tried to design the band reject filter using FR 4 substrate with thickness of 0.8 mm and with a  $\lambda$  by 4 resonators. In the next lecture we will extend this and we will say how  $\lambda$  by 4 resonator will act like a band pass filter and after that we will discuss about the design of power divider and hybrid coupler.

Thank you very much.