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

**Module – 12**
**Lecture – 59**
**Mixer Design in NI AWR Software – I**

Hello welcome to the last second session in the series of Software Demonstrations. In this lecture we are going to study how to use computer aided design tools for the design of solid state microwave circuits. I will demonstrate this with the help of a mixer design example. Let us begin.

(Refer Slide Time: 00:36)



So, the software platform or the cad tool that we are going to use in this particular is national instruments AWR, design environment platform. This platform consists of 3 different tools, the first one is microwave office which is used for RF and microwave circuit design The second one is visual system simulator which is mainly used for system level simulation and third is analogue office which is typically used for analogue RFIC designs. All these 3 tools have facility of linear and non-linear circuit simulations, EM simulations, and circuit and system level co-simulations.

In this lecture we are going to focus on microwave office and we are going to use linear and non-linear circuit simulators available in the microwave office.

(Refer Slide Time: 01:25)



So, let us revise for given specifications first we choose the mixing device that we are going to use, after that we choose the type of the circuit that we are going to use, and then we design the circuit which involves design in various components involved in the circuit, then we have simulation and finally, we optimize a circuit to achieve the desired performance.
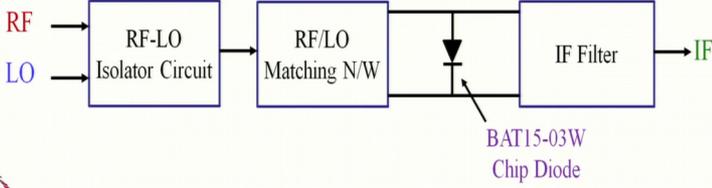
(Refer Slide Time: 01:50)



In this example we are going to use diode as the mixing device and the circuit we have chosen is simple single diode mixer. The specifications are we have an RF frequency at

4.25 gigahertz we have the desired if at 500 megahertz which is 0.5 gigahertz. And depending on this RF and IF specifications the LO which is local oscillator frequency chosen is 3.75 gigahertz.

The circuit is as follows as we can see there is an RF LO isolator circuit to isolate the RF and LO ports, then we have the matching circuit this is the mixing device and we have an IF filter. The mixing device which is the diode is nothing, but a Schottky diode the part number is bat 15-03 W. This is a Infineon diode of chip form which can be used in microstrip design.

(Refer Slide Time: 02:48)
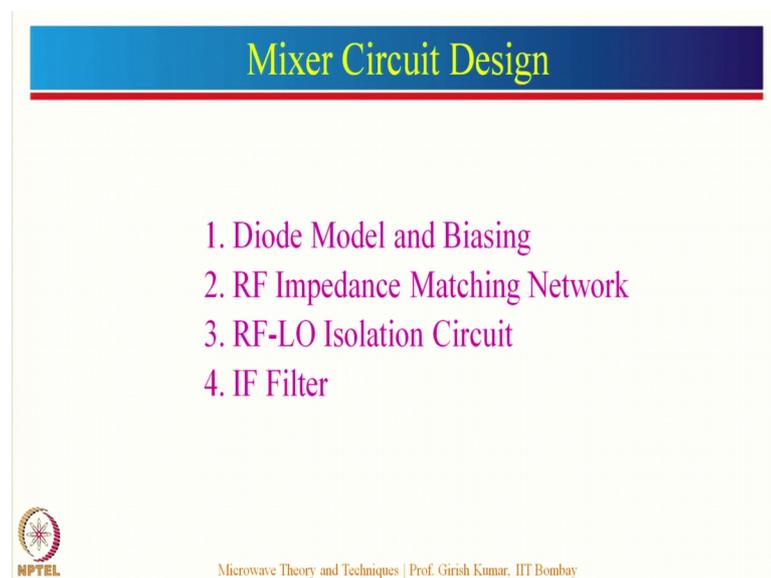


The steps or the sequence in the design that we are going to follow is this. So, first is this diode we are going to model this diode in this platform, after that we see the input reflection coefficient at the RF frequency, and based on that we will design the matching network, after that we will design the RF and LO isolation circuit and finally, the result of this we will simulate we will see the spectrum output at the output of the mixer. And then accordingly we will design the if filter to filter out the undesired frequency components and finally, we will get the if output.

(Refer Slide Time: 03:23)
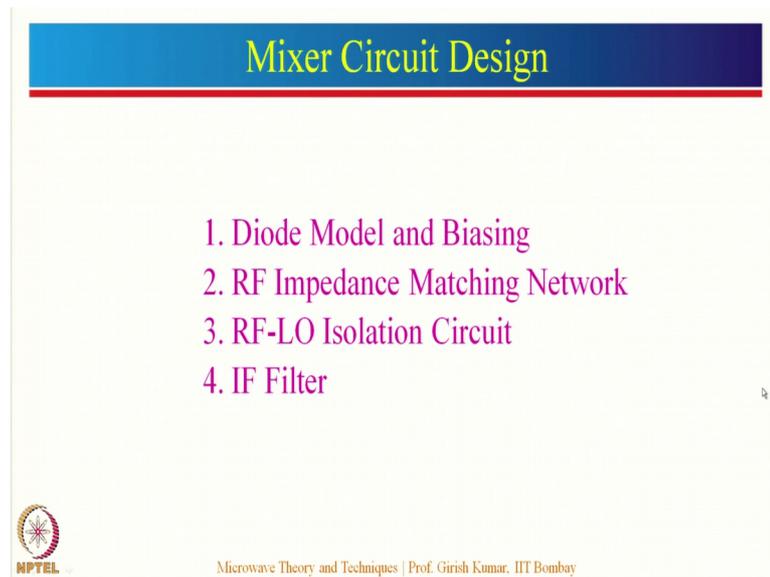


Once all this is done we will simulate the entire circuit to get the conversion loss performance, the isolation performance and noise figure performance of the mixer.

(Refer Slide Time: 03:26)
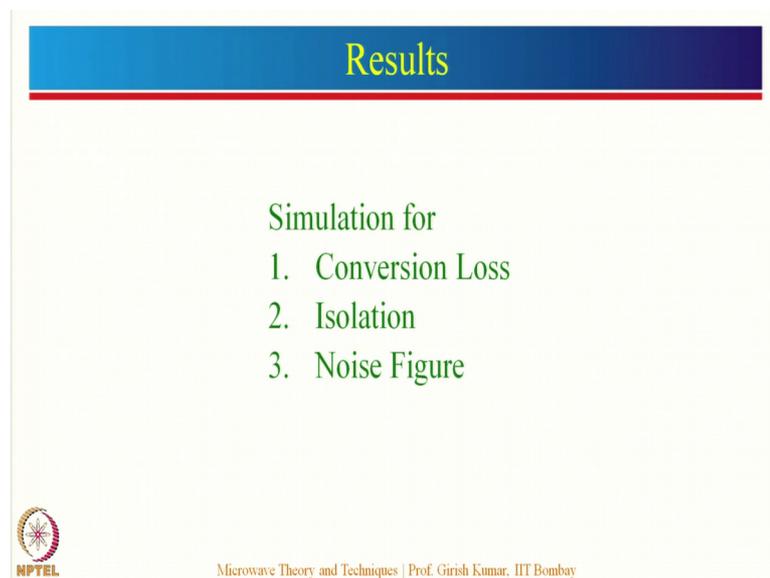


Let us move to the software part, whenever you open the NI AWR software the user interface looks like this.
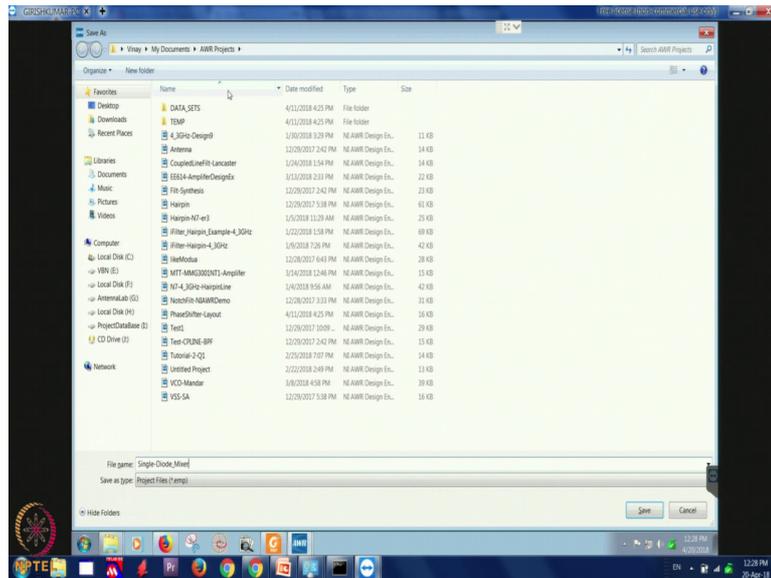
(Refer Slide Time: 03:44)
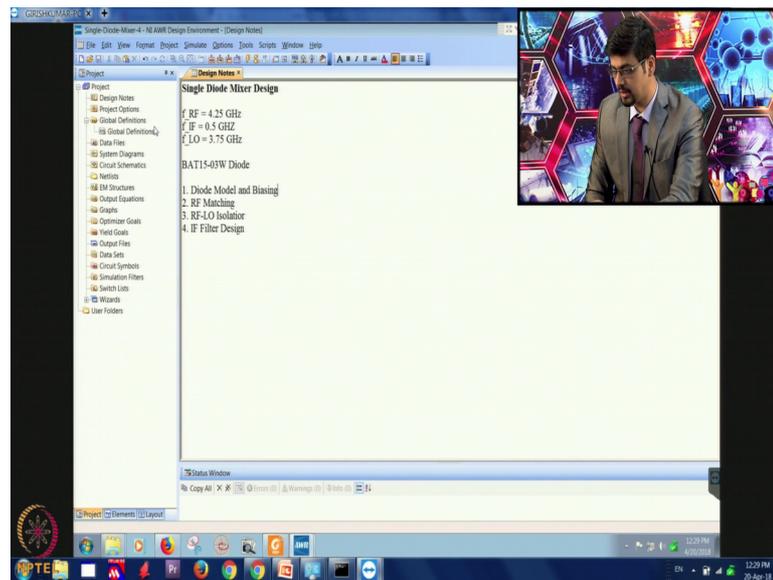


At the top you have the project name which is currently untitled project because we have not saved the project yet, you have couple of toolbars and on the left you can see you have 3 browsers - one is the project browser, another one is a element browser and the third one is the layout browser. So, the project browser consists of various things first you have design notes, you can have project options, you have global definitions, data files, system circuit diagrams, graphs and so on.

So, let us first start with the design notes, you can double click on it and you see that file will be opened, you can type in whatever you want in this file. So, let us say we start with single diode mixer design f RF is 4.25 gigahertz, f IF is 0.5 gigahertz, f LO is 3.75 gigahertz. Now, you can type in whatever you want in the design notes and we will first save the project. We will give appropriate name single diode mixer, I save this here. So, the project is now saved.

(Refer Slide Time: 05:02)



So, first thing we are going to build a diode model. So, go to circuit schematics right click new schematic, name this as diode model.
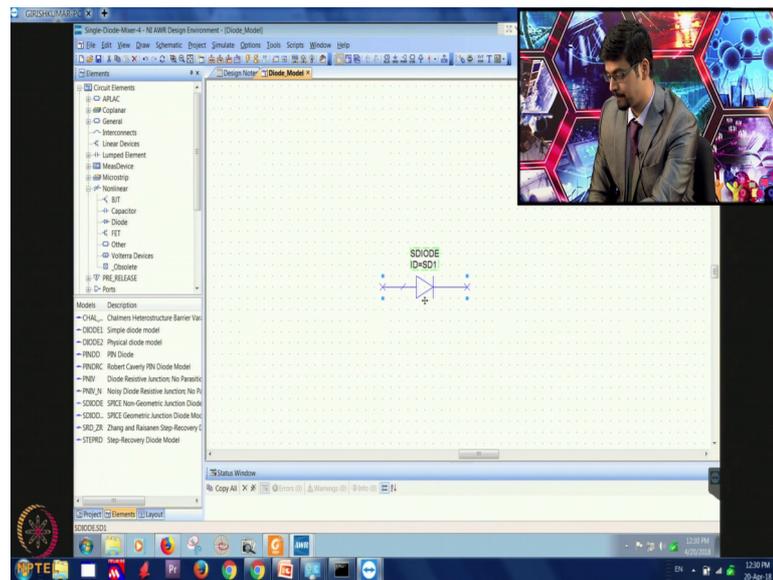
(Refer Slide Time: 05:32)
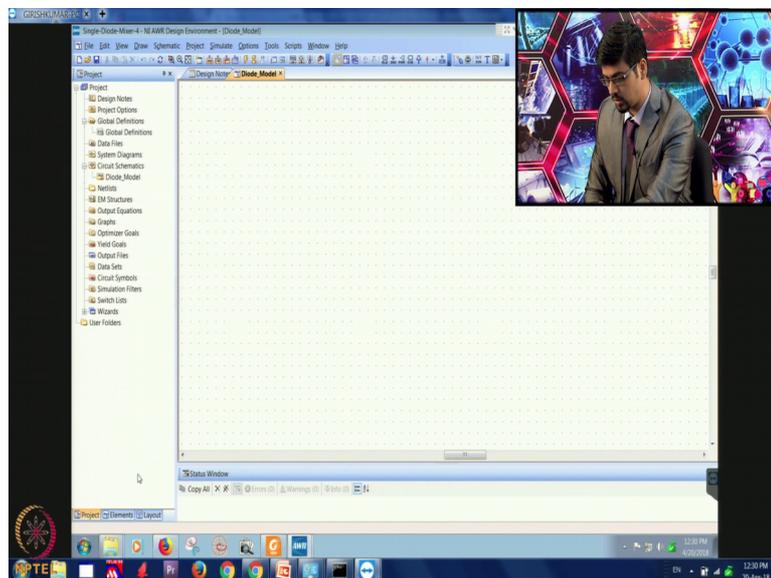


In this we are going to import first an element, go to non-linear diode. If you drag this you can read the description.

(Refer Slide Time: 05:37)



We are going to use s diode which is spice non geometric junction diode model, just drag this into the circuit schematic window, click it, it will be placed.

(Refer Slide Time: 06:06)
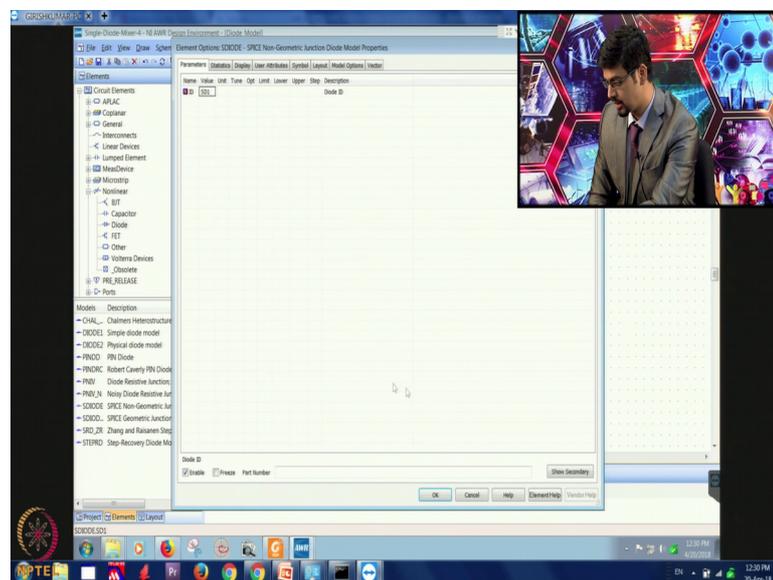


Now, if you double click on this you will see the options related to this, right now you do not see anything. You have to click on show secondary. And you will see all the diode parameters which are basically diode junction parameters which are specific to a particular diode.

(Refer Slide Time: 06:10)
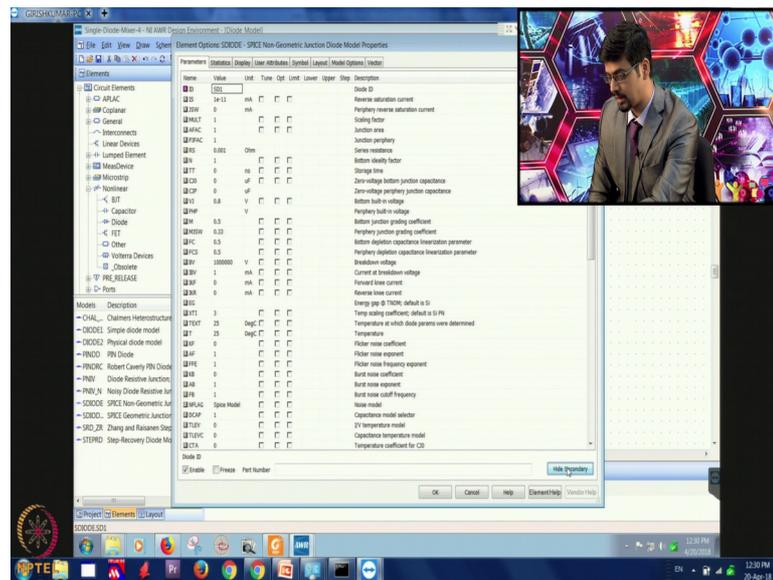


Now, we will fill this parameters based on the device which we have chosen. The manufacturer provides the data sheet of this diode or you have spice models of such devices available on internet.

(Refer Slide Time: 06:37)
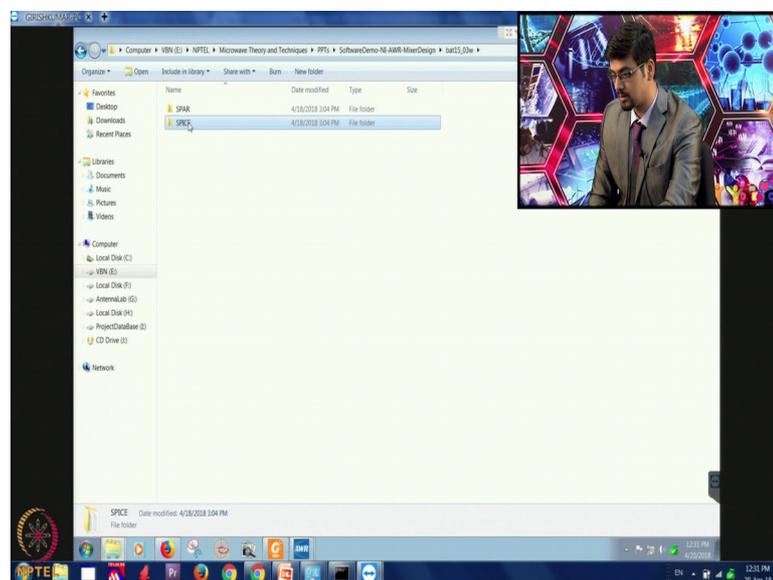


So, we have the spice model available with us we will choose that.

(Refer Slide Time: 06:40)



This is the spice model file and as you can see there are different specifications which are mentioned over here. So, IS reverse saturation current is 74 nanoamperes, N is 1.07, RS which is the series resistance is 5 ohms and so on. We will fill the data using this file into our model.

Now, this diode is of chip form so it has a package associated with it and the package will have some parasitic inductance and capacitance values. And the spice model also contains these parasitic values and we will include these values in the diode model. So, SOD 323 is the package.

(Refer Slide Time: 07:30)



And this file contains the parasitic. So, this is your chip which is diode and you see that the inductance in series a capacitance in parallel and you have two lead inductances. So, we will build this model in our file. We will go to elements, we will go to lumped, we will have inductor, we will choose inductor closed form, drag, place it here, we will go to capacitor, capacitor closed form, we will place it here, just zoom out a bit and we have two more inductances at here and at the output port.

(Refer Slide Time: 07:49)
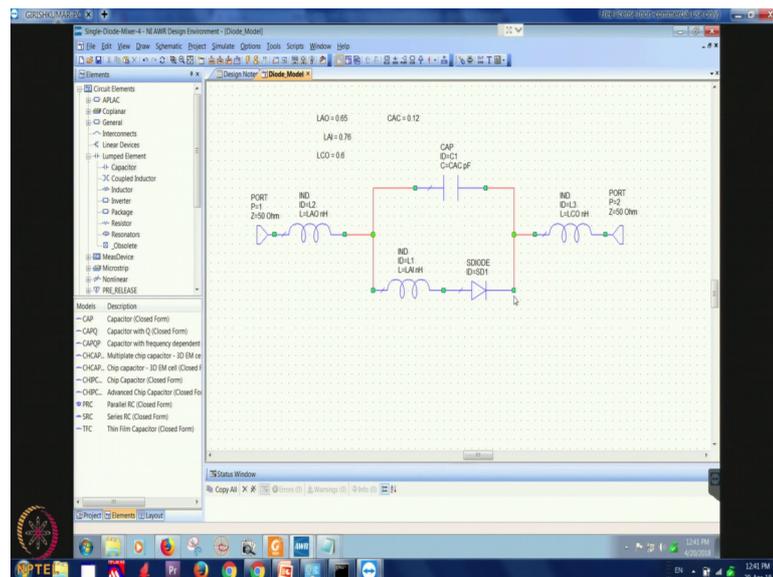
Now, the values of this inductors and capacitor have to be specified. So, instead of specifying the values directly what I am going do is to create variables. So, for example, this inductance has a name LAI. So, I will place this value you just have to double click on that particular L. So, LAI, I specify it as LAI. Then this is LAO, this is LAO right and as you can see all these things are in red because I have not specified the values to these variables. So, click on here, click on equation, place it here and then specify LAO is equal to LAO is 0.65 nano, 0.65 no need to specify nano as it is already specified here.

Now, I will connect this for connections just move your mouse to the unconnected part, a symbol will appear just left click and go to the next part, click again a wire will automatically be connected. So, I will do the connections like this. At the two ends of these inductors what I am going to put are the ports. So, just go to port and place a port here take another port, right click to rotate, place it here. So, you have port 1 at this end port 2 at this end. The reason for putting the ports will be clear after sometime.
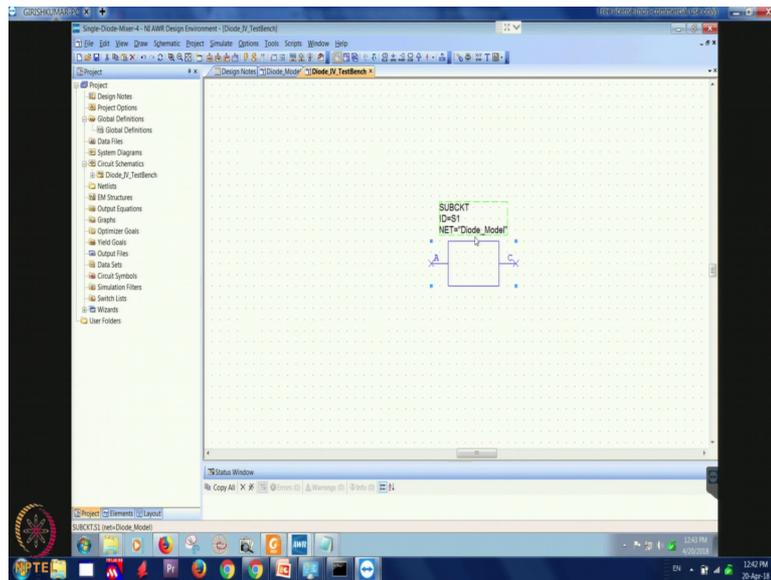
(Refer Slide Time: 10:30)
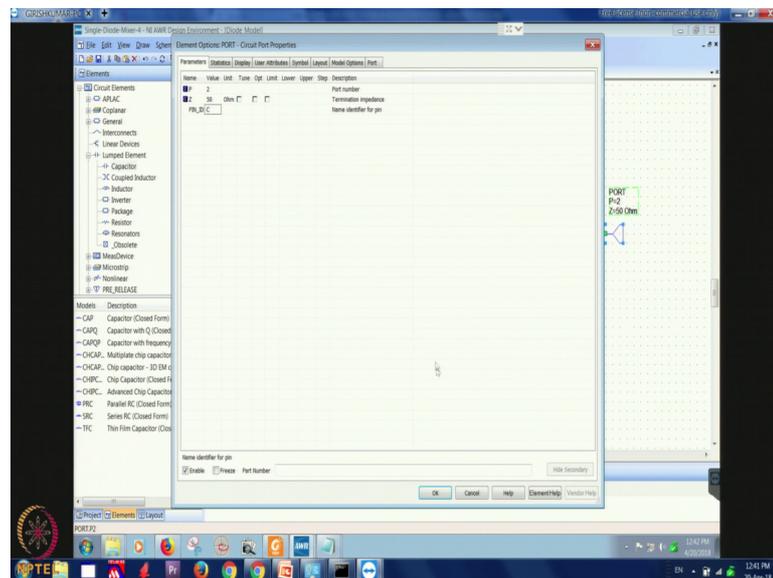


Now, this terminal is the anode of the diode, this is a cathode so the port 1 corresponds to anode I will create a symbol saying A and for this I will say the pin ID as C. So, anode and cathode, I save the project.

Now, next point is we want to test this model of the diode and we will see that with the help of the IV curve of the diode it should match with the data sheet specifications. Go back to project browser, circuit schematics, right click, new schematic, diode IV

testbench is the name that we will give to this file. And in this file now I want to include whatever circuit I had in the diode model. What I will do, I will create a sub circuit diode model is already there, normal selected hit, and you will see that now the entire circuit which has been designed here is now taken as a black box.
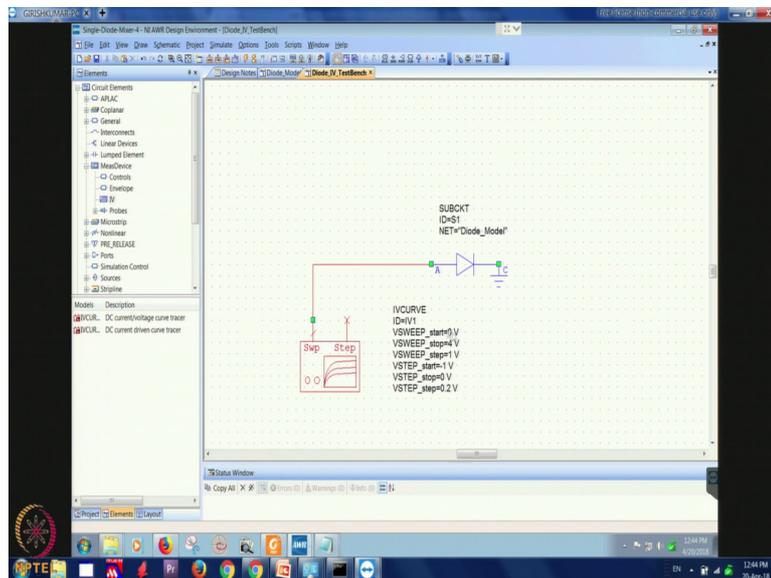
(Refer Slide Time: 11:33)



Now, this symbol can be changed just double click on, it go to symbols and you can choose any symbol that you want we will choose a diode IC symbol for this because it represents a diode IC.
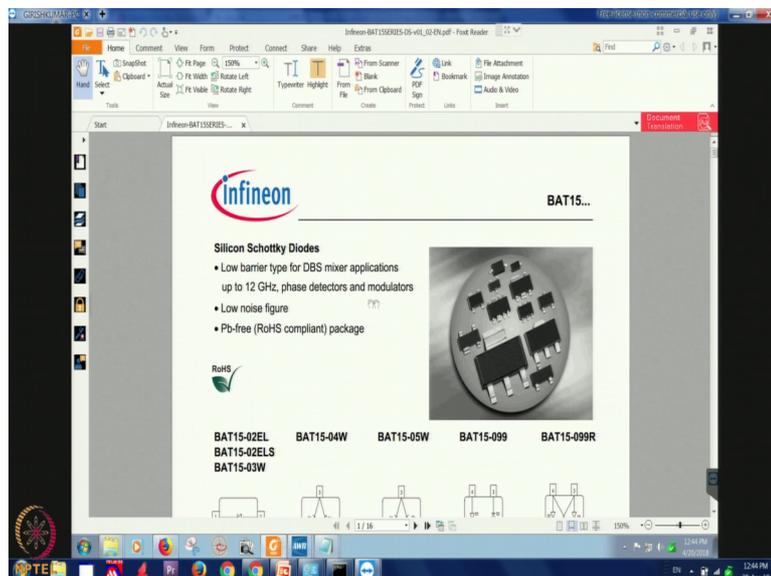
(Refer Slide Time: 11:50)
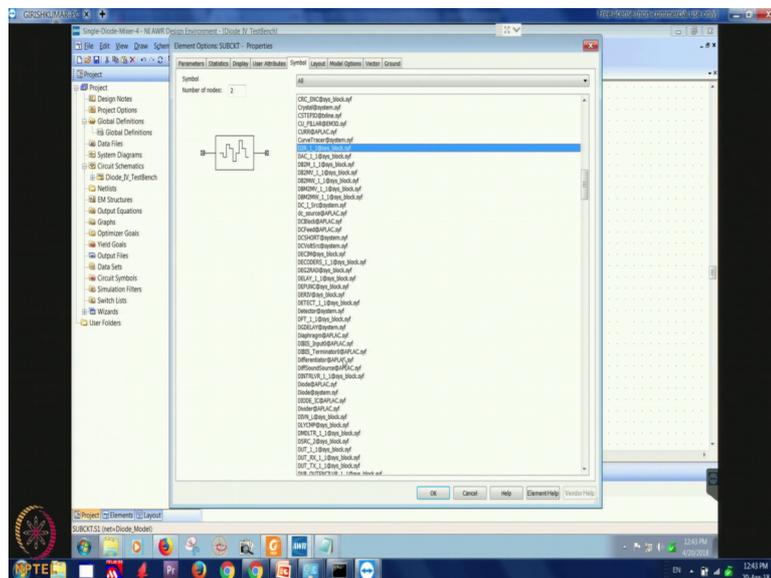
Now, this is anode and cathode of this diode.

(Refer Slide Time: 12:14)



Now, simple to measure the IV curve I will ground the cathode at the anode I want to provide a voltage sweep. For that I will go to elements, I will go to measurement devices, I will go to IV, and I will take a DC current slash voltage curve tracer, I will put it here, I will just drag this here, I will connect this to anode. This is a sweep port the V sweep ranges from 0 to 4. Now, it is worth looking at the data sheet of the diode to check what voltage range we should have. So, let us have a look at that data sheet.
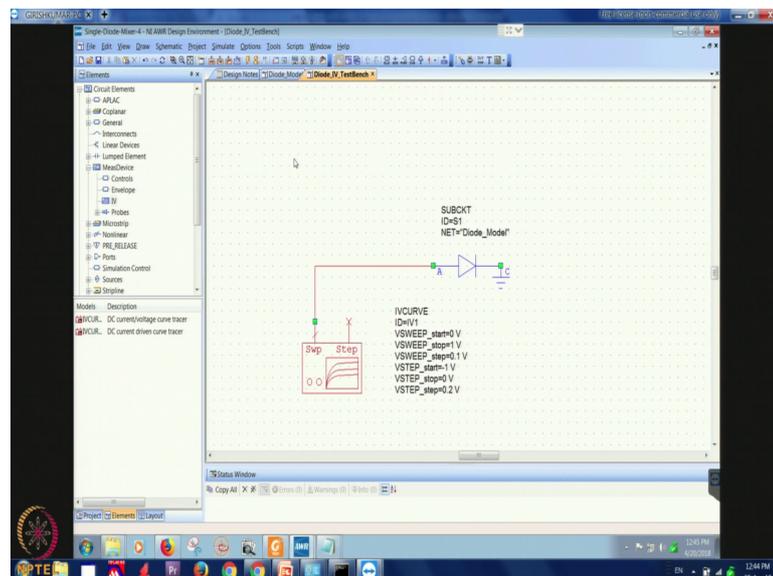
(Refer Slide Time: 13:03)

This is diode that we have selected and we see that in the curve the voltage variation is from 0 to 1 volt.

(Refer Slide Time: 13:14)
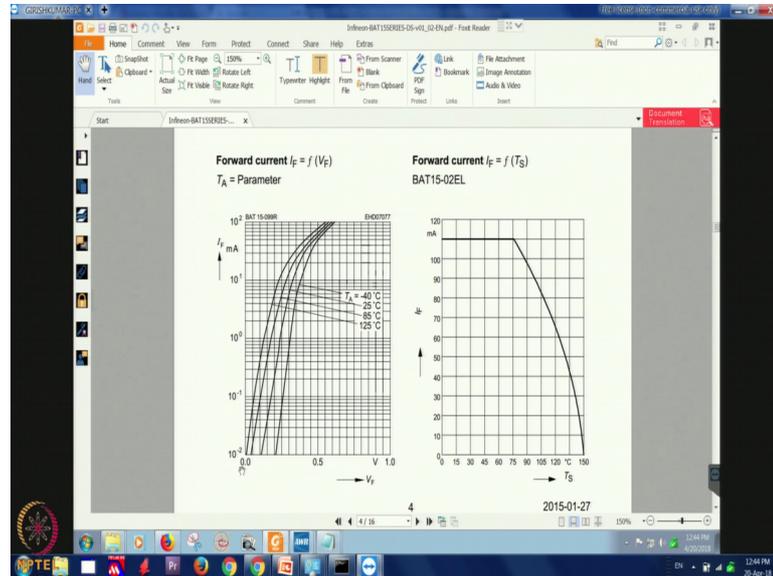


So, we will stick to that range so 0 to 1 with the step of 0.1 let us say and my circuit testbench to measure the IV characteristic of this diode is complete.

(Refer Slide Time: 13:23)



Now, I will create a new graph. So, click on this add a new graph diode IV is a name I will give to this graph and let it be of rectangular type create you will see a blank graph here.
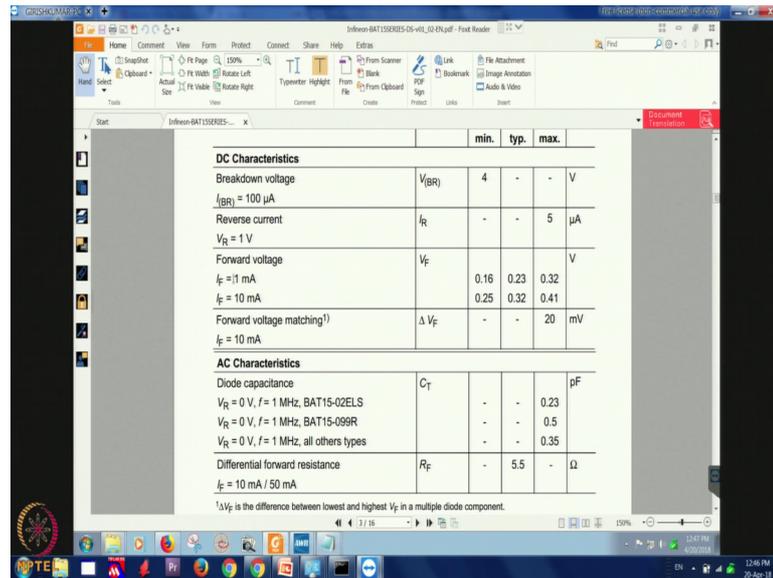
(Refer Slide Time: 13:47)
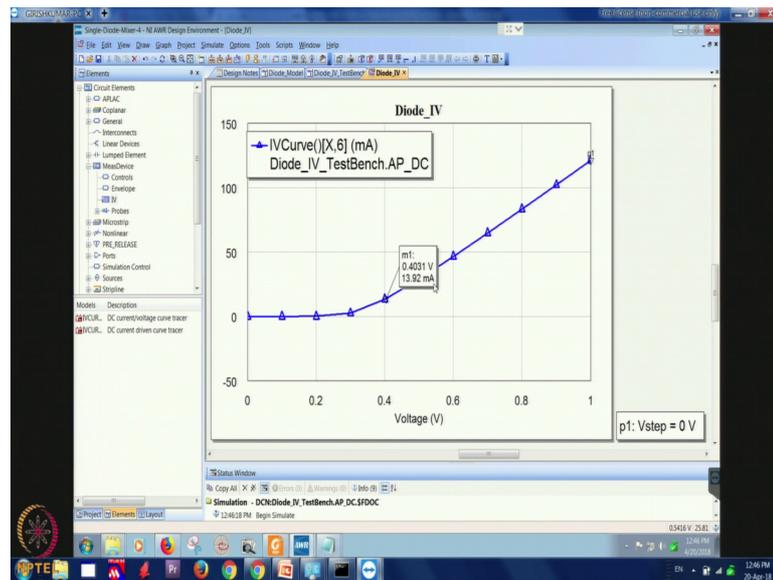


Now, we have to add measurement to this graph, right click, add measurement, go to non-linear current, select IV curve data source there are various options and we want to choose diode IV testbench, and IV curve dot IV 1 sweep which is the variable that we are going to use is used for the x axis and the step is being ignored.

So, we will just choose a simple 0 volt for the step and the simulator is APLAC DC and it apply, ok. To simulate we use this button which is analyze and we see that the diode IV curves appear. I will just drag this particular thing right and I check the values of diode currents and voltages right click, add marker click on the graph and you see that 0.4031 is the voltage across the diode and 13.92 is the current flowing through the diode. Now, to verify that we are getting the correct results we will use the DC characteristic of the diode which is specified over here.

(Refer Slide Time: 15:21)



So, for forward voltage V F with different I F which is the diode current. So, for 1 milliampere you should get the forward voltage in the range of 0.16 to 0.32 volts, we will check that. So, to move this marker to 1 milliampere, right click, marker search value of one find y value which is the current and search towards the left search. So, for 1 milliampere the voltage is 0.2366 volt. In the data sheet you have nearly the same value which is 0.23 volt. Similarly for value of 10 milliampere I will search right, you see that the voltage is 0.3685 volt which is in the same range. So, this confirms that whatever diode model that we have built is in sync with the diode data sheet provided by the manufacturer, cancel this.

Now, we will move to diode biasing project, circuit schematics, new schematic name it as diode bias, create. Again I will add the sub circuit. So, instead of going to the circuit again I will just copy this, I will paste here, right click to rotate anode cathode, I will ground the cathode, moving this a bit.
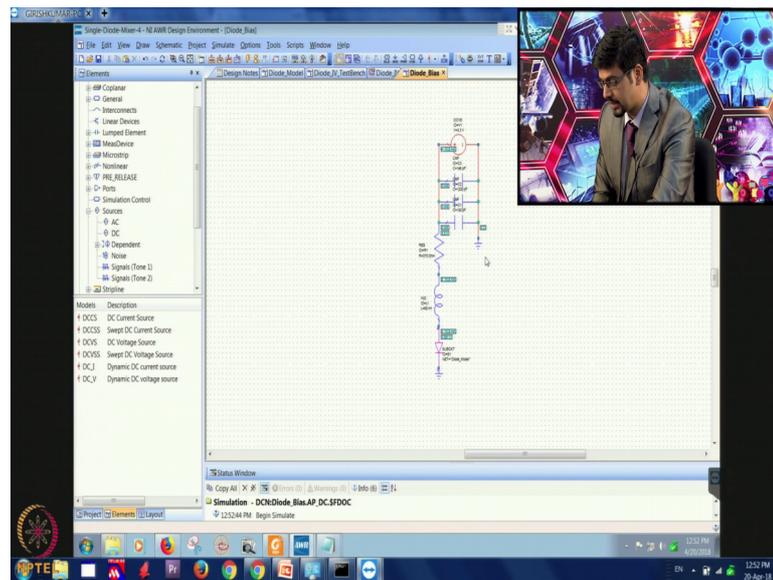
Now, for biasing I need a RF choke to avoid the RF currents to flow into the DC path, so first I will take an inductor go to lumped inductor closed form. Then I will have to take a resistor to limit the current that is flowing into the diode, I will place a resistor over here, then in between this resistor and the supply voltage I have to put capacitors to remove the transients and high frequency noise. So, I will go to capacitors, I will choose capacitor, I will place one here copy paste, I will place the second here paste, third here.

And then I have to choose a DC voltage supply. So, I will go to sources which is this DC and I will select a DC voltage source, drag it and you can right click rotate and place it in this fashion. We are going to use supply voltage of 3.3 we will first complete the circuit we will place a ground we will connect it. We will connect this again and based on the diode IV curve what we are going to use is this particular biasing point, so current of 10 milliamperes and voltage of 0.368 which is around 0.37 volt. The resistor which is closed by giving these values is around 270 ohms based on the calculations. So, we will put the resistor of value of 270 ohms.

The inductance value for avoiding RF current is 68 nano Henry, 68 nano and the capacitances to bypass the noise and transience is this is 100 picofarad this is 1.2 nano and this is around 1 micro. So, 1E6 pico right and the voltage is 3.3. Now, to see that appropriate voltages and currents are set across the diode we are going to use this tool called annotate add new annotation to the circuit.

(Refer Slide Time: 20:15)
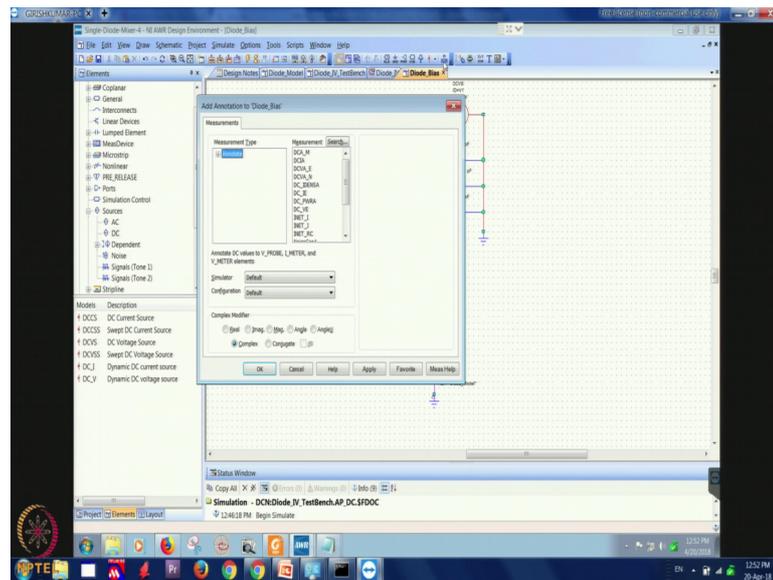


So, we are going to use DCIA which is annotate DC input current for all elements the circuit is diode bias current apply, and you have DCVA apply.

(Refer Slide Time: 20:47)
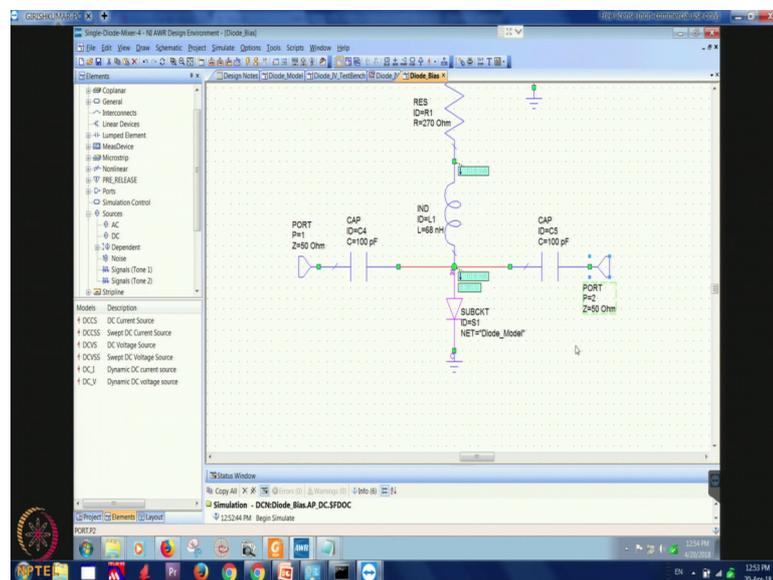


Then if you simulate you see that the current that is flowing into the diode is around 10.8 milliamperes and the voltage set across the diode is 0.38 volts which is quite close to the desired biasing point. This confirms that our biasing is correct.

Now, to add circuit around this diode we need to avoid DC current flowing into the other circuit elements. So, what we will do? We will add DC block capacitors at the input port

and as well as at the output side, we will connect it. The value of this capacitor is approximately 100 picofarad which will pass the RF current, but block the DC. Now, this is the input of the diode this is the output I will place ports, port 1 at this end port 2 at this end. This is the input so I will create an ID for it IN, this is the output. So, I will create an ID for that which is OUT and we are all set to apply high frequency signals input to this diode, output will be taken at port 2. So, this is our complete diode bias circuit.

The next step what we are going to do we will see the RF input impedance of this diode and accordingly we will design the RF matching network. For that again create a new schematic, right click, new schematic I will name it as RF match, testbench create. And we are going to use this diode bias circuit as a sub circuit, diode bias, as you can see this is the IN this is the OUT.

(Refer Slide Time: 23:10)
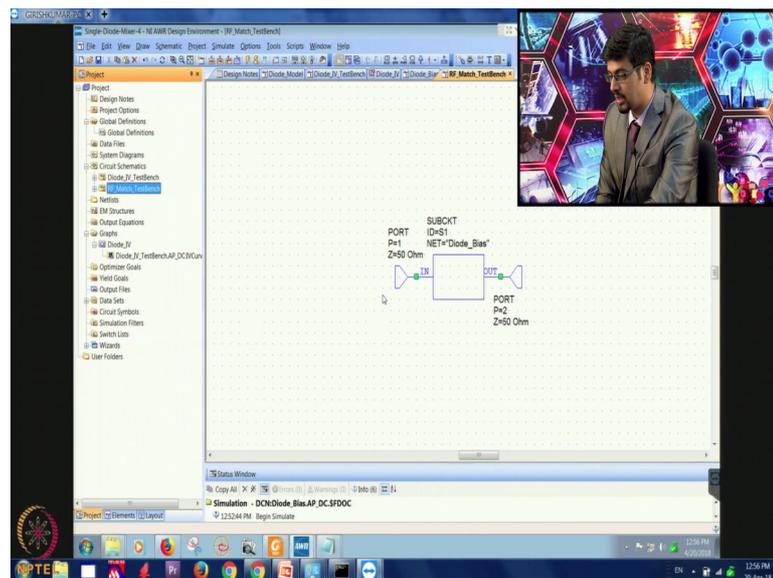


Now, simply put ports at the IN and OUT ports of the diode, and we are interested in looking at the s 11 at port 1 provided at the RF frequency. To set the frequency go to the current schematic which is RF match test bench, right click on it go to options uncheck use project defaults, we want a single point operation because currently the RF frequency is 4.25 gigahertz and hit apply. So, the circuit will be analyze only at 4.25 gigahertz.

We want to plot the s 11 on the smith chart. So, create a new graph, the type should be smith chart, name I will say s 11 RF.

(Refer Slide Time: 24:28)
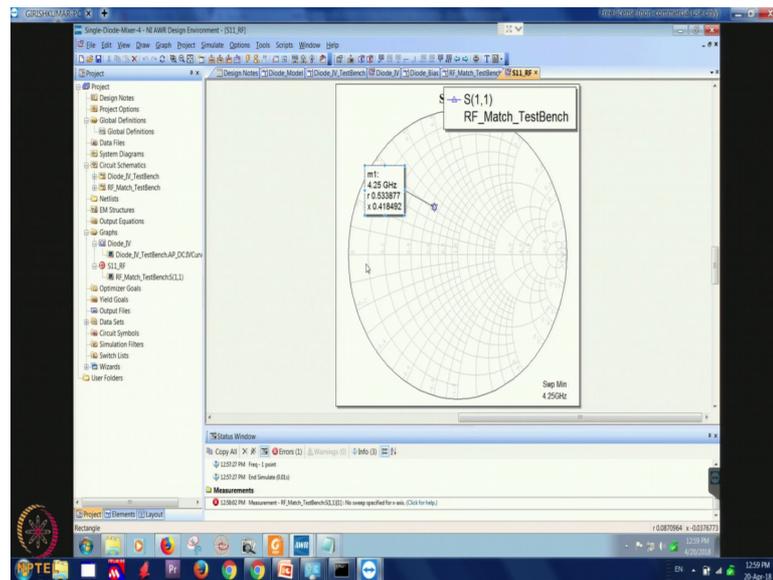


Create this is the smith chart to add measurements right click, add new measurement, go to linear port parameters, S-parameters, choose appropriate circuit which is RF match testbench S 11 we are going to see and here select use for x axis, ok. Now, you see that a marker is added over here if you right click add marker click on it, you will see that the frequency is 4.25 gigahertz the normalize r is 0.5 3 and normalize x is 0.41. So, the technique that we are going to use or the circuit that we are going to use in this particular case is single stub matching technique. For that you know the procedure.
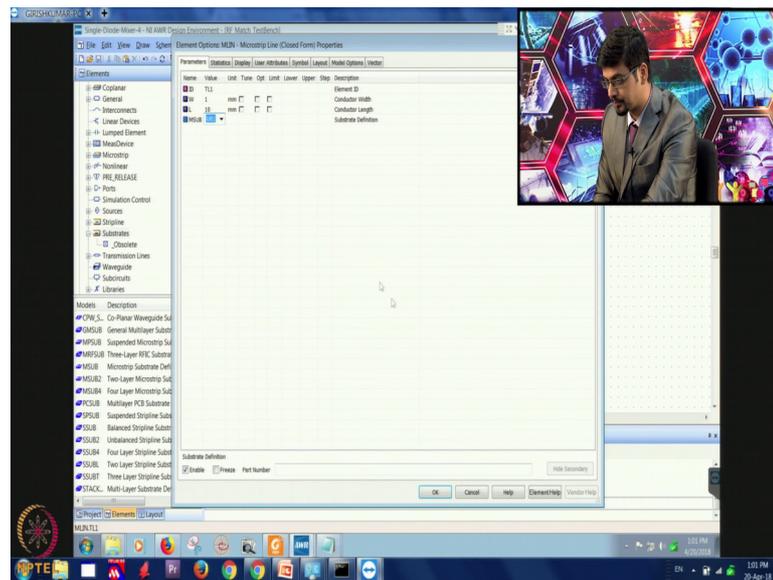
(Refer Slide Time: 25:21)
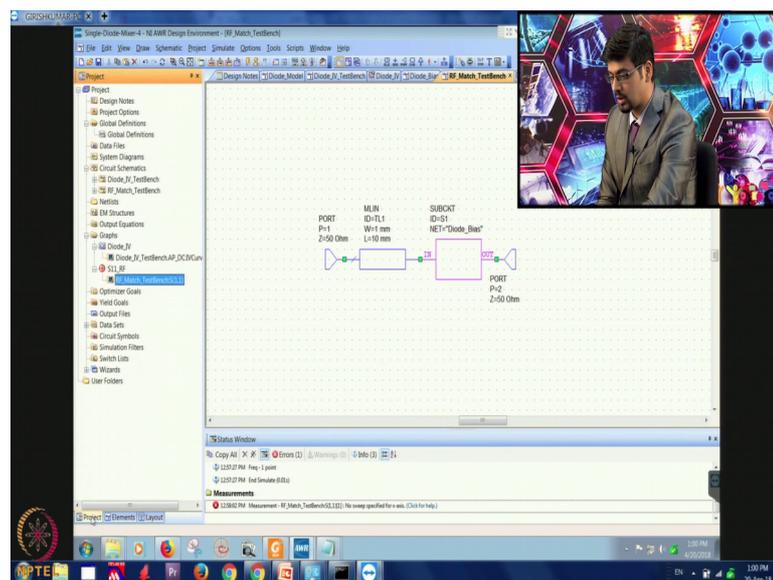
So, first we have a series microstrip line in single stub matching. So, go to elements microstrip lines, MLIN, drag and place it over here. Now, this microstrip line has to have some substrate definition, and because a single substrate will be created for all the components being used in the circuit will go to global definitions and we will add a microstrip substrate definition at this particular point. So, for that go to elements, go to substrates, which is here click on it and choose MSUB, drag it here, place it.

(Refer Slide Time: 26:20)
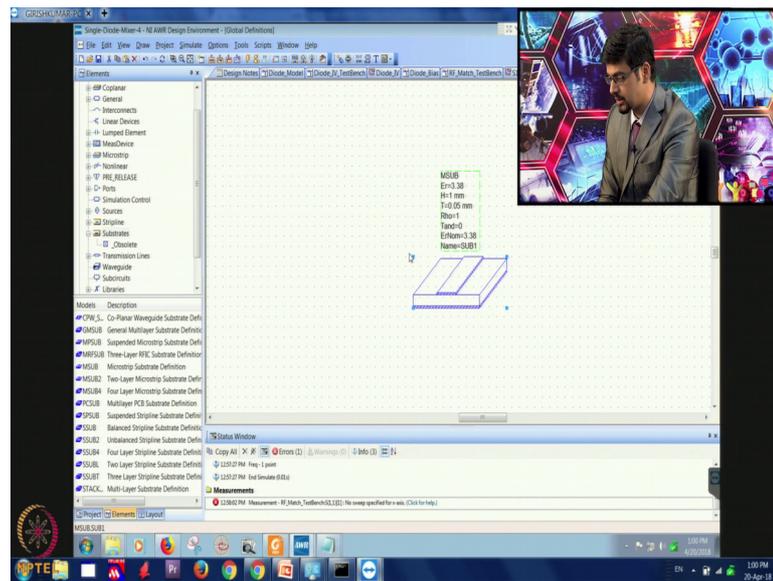


We are going to use an RT duroid substrate so 2.2, H is 0.787 mm, thickness of the conductor will be 0.035 mm which is 35 micron, tan delta is going to be 0.0007 and this is 2.2 and the name of the substrate is sub 1. Save it, we can close this, go back to our design, right click on it, go to properties MSUB. You can set it to sub 1, ok.

(Refer Slide Time: 26:59)



Now, the width and the length of this line which is a part of the matching network. We all know that the width should be corresponding to the characteristic impedance of 50 ohm.

(Refer Slide Time: 27:11)
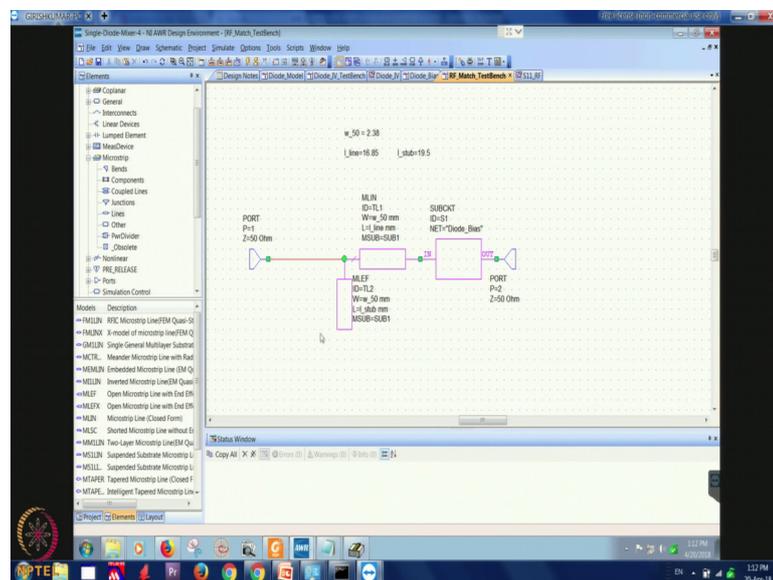


So, I will set it to w 50 which is a variable and length I will set it to l underscore line. I will have these two equations, so w 50 is the width corresponding to the substrate that we just defined and the characteristic impedance of 50 ohm.
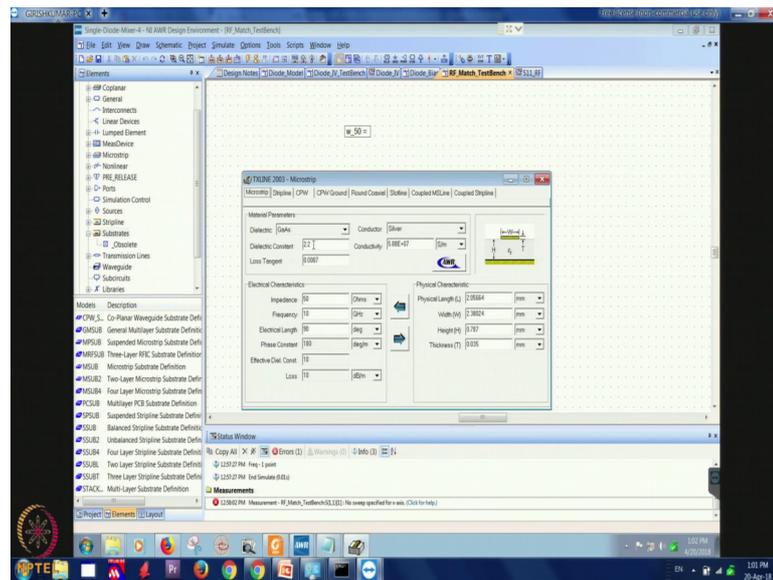
Now, to calculate this width we will use a tool go to tools TX line the substrate details you can enter here 2.2, loss tangent 0.0007 and the copper should be selected as the conductor. The substrate height is 0.787, thickness is 30 micron, the impedance required is 50 ohm, the frequency is 4.25 gigahertz, and currently the length is not of a concern we are concerned about the width. So, just click on this arrow and you will see that the width is 2.38 mm. So, I will add w 50 equal to 2.38 l line has been frozen to 16.85 mm.

Next, task I want to add a stub. So, I will again drag it, I will go to microstrip, I will go to lines, this is the open circuited stub, we will drag it, rotate and we will place it here, and again I will connect it like this. Again the stub has a width corresponding to a 50 ohm characteristic impedance. So, I will set it to w 50 which is already defined to 2.8, 2.38 mm; l I will again create a variable l stub, the stub length is frozen to 19.5. So, a line length of 16.85 mm and stub length open circuited stub length of 19.5 mm will give me desired match which is at the centre of the smith chart. So, this is my independent matching network. We will stop at this point. We will study rest of the circuit in the next lecture.

Thank you.