

Introductory Neuroscience and Neuro-Instrumentation
Indian Institute of Science, Bengaluru
Lecture 64
Stress Tissue Analysis Using COMSOL Multiphysics

Welcome to the session, we are going to talk today about Stress Tissue Analysis. When we go for ECG measurement for any specialize doctor, we usually have to use wet electrodes and the new technology because of the issues of using the wet electrode now is moving towards dry electrode. And when you have, when you use the dry electrode as you can see over here, it involves very pointy electrodes and they are like multiple electrodes as you can see over here.

And you need to press it on your scalp and it presses a little bit and if you have a design like a headband and in that case, the pain that is seen by the person who is coming for this ECG is very difficult to get quantified. So, one of the way how people are using to quantify the pain is to perform a simulation on different types of electrodes and how it is creating different level of stress on the different parts of tissue which has different Young's modulus or density.

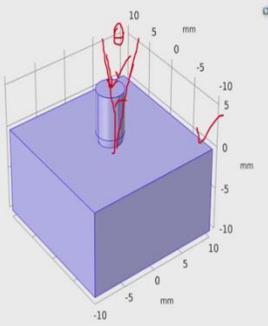
So, today we will learn on how to actually model the stress analysis of the tissue when an electrode is pressing on top of it. So let us go to the presentation.

(Refer Slide Time: 01:50)

Model Setup

- The Probe was modelled as an linear elastic material model
- The Tissue was modelled using the material properties of the tissue.

	Young's Modulus [Pa]	Poisson's Ratio	Density [kg/m ³]
✓ Polysilicon probe	1.69e11	0.22	2320
✓ Tissue	1e4	0.49	1080



The image shows a 3D model of a probe (a red cylindrical structure with a sharp tip) pressing down on a blue rectangular block representing tissue. The probe is positioned on top of the tissue, and the tissue is deformed under the pressure. The axes are labeled in millimeters (mm), with the vertical axis ranging from -10 to 10 and the horizontal axes ranging from -10 to 10.

So, this is what we are trying to model. So on the top is the probe as you can see over here and in the bottom, we have the tissue. What we are going to do over here is we are going to apply a

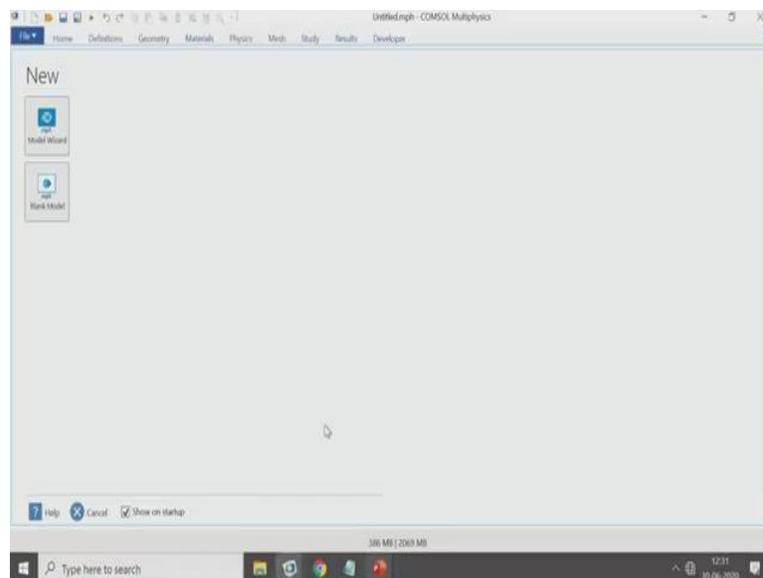
particular pressure on the top of the probe, so this is the dry electrode and we are going to apply a pressure of around 100 Pascals on the top and what we are going to also do is we are going to indent the probe with a particular angle.

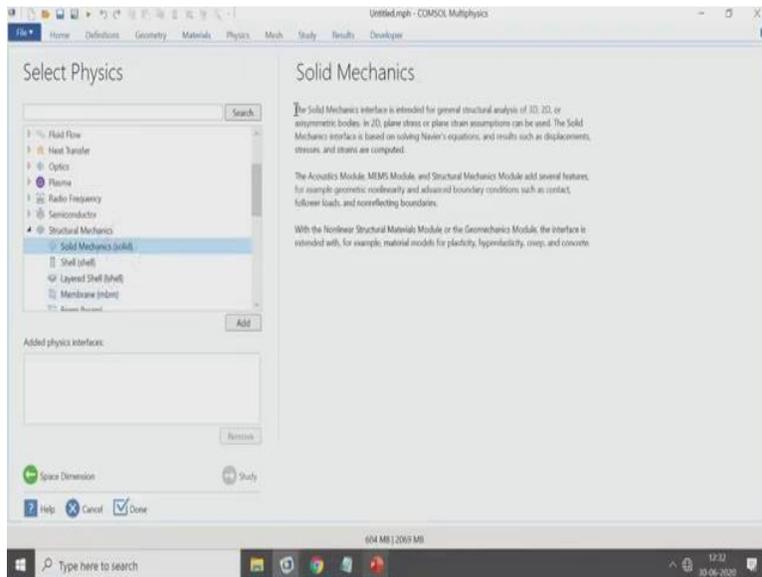
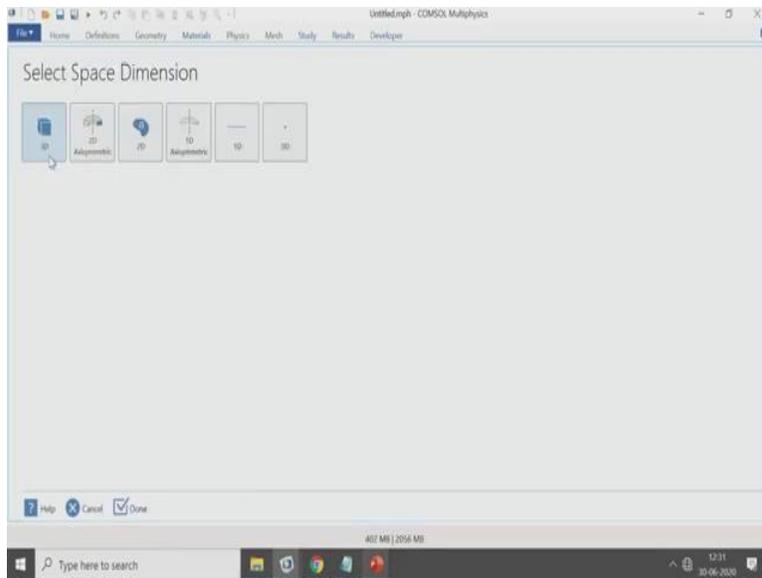
So, we are going to do a sweep of this angle and try to see how the stress below this tissue at a certain distance, how it is wearing. So which angle is better, which angle gives more stress and the stress profile in the tissue of different-different angles and pressure. If we know the threshold of the stress value which can cause the pain with that you can get to know how tight the headband should be for example, or what angle of inclination the probe should be.

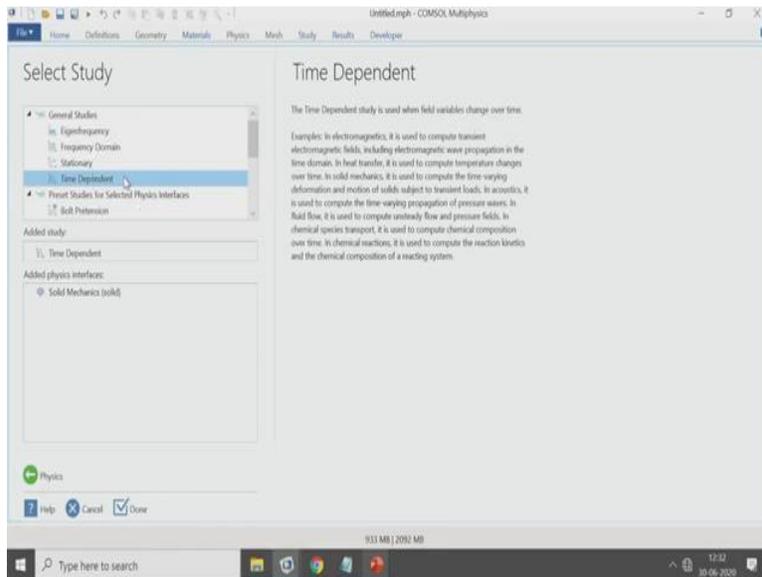
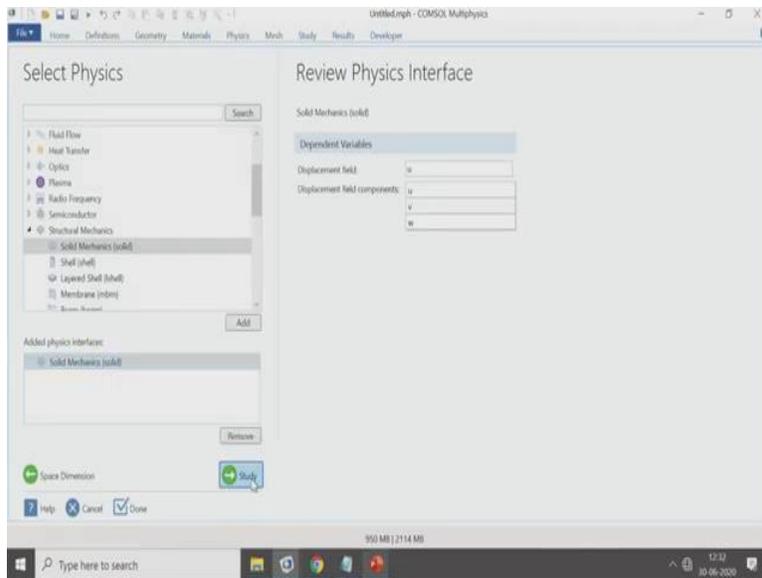
To model this we first need to know the material properties of both the domain that we are modeling. For example, you have the first one is the probe electrode, this electrode properties has been mentioned over here. So in this case, we have used a very simple polymer material that is poly-silicon. In other cases, you can use metallic probes. In those cases, the Young's modulus, Poisson's ratio and density will change.

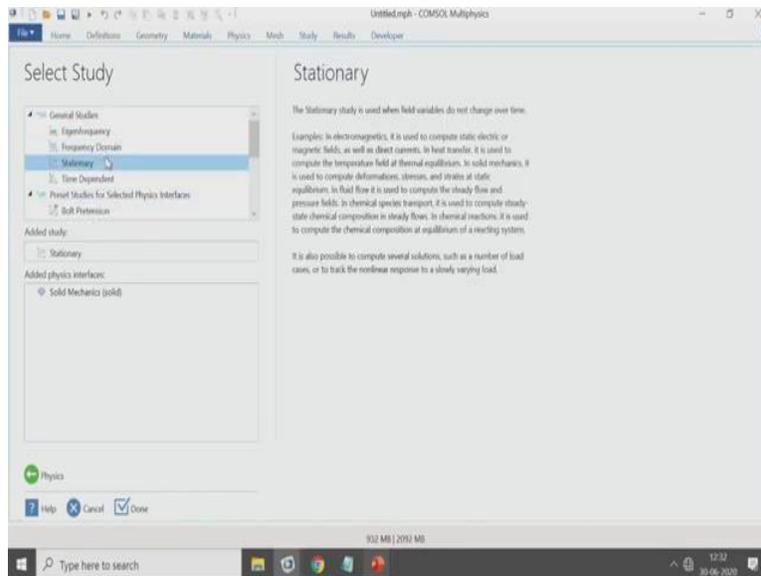
We are going to also use the tissue properties which are easily available if we do a literature survey. So let us go into COMSOL. So today we are going to use COMSOL to understand how the stress profile within the tissue varies under the effect of force by the electrode.

(Refer Slide Time: 03:57)





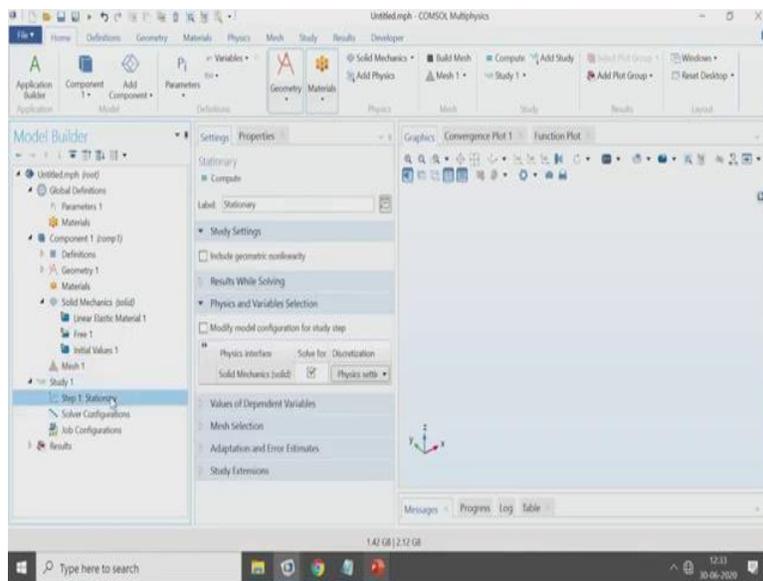
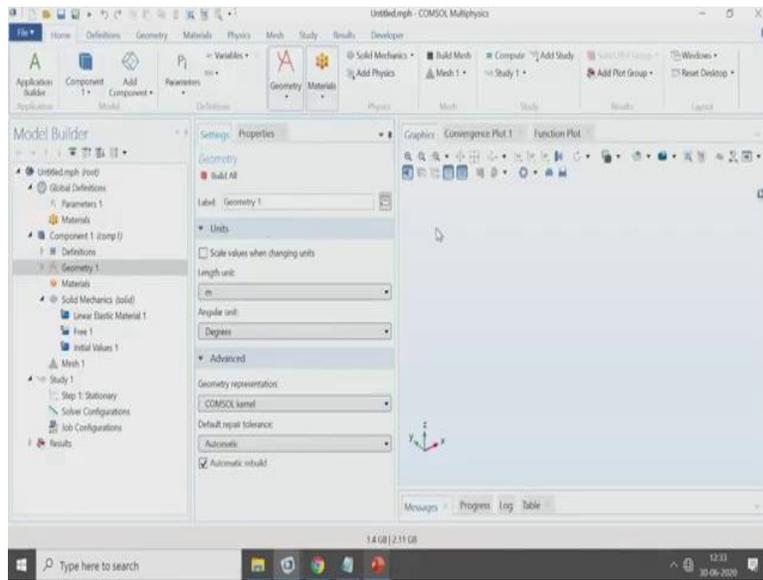


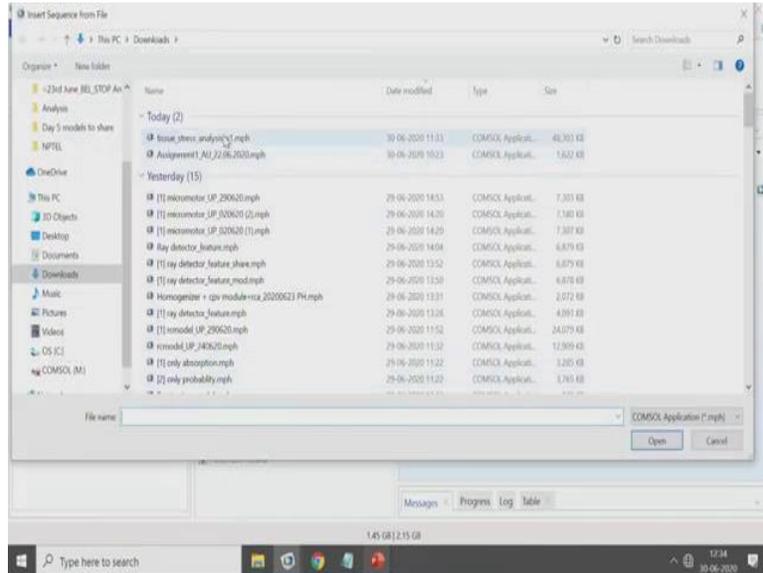
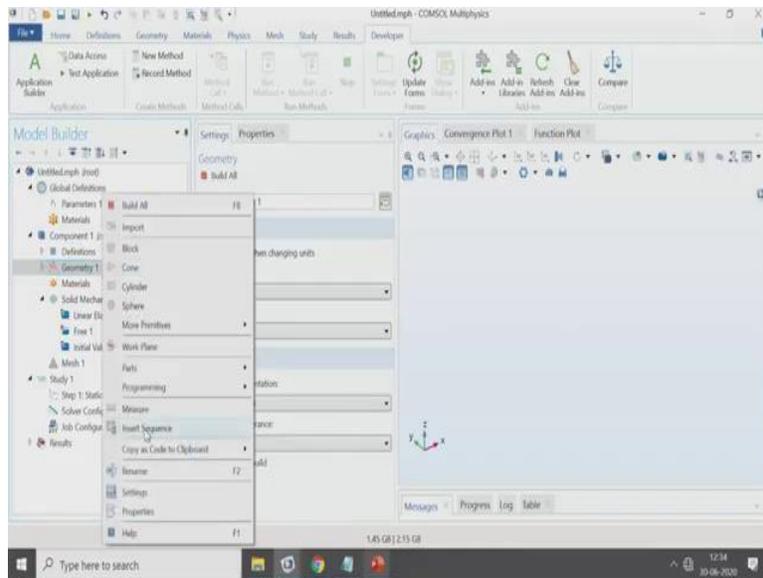


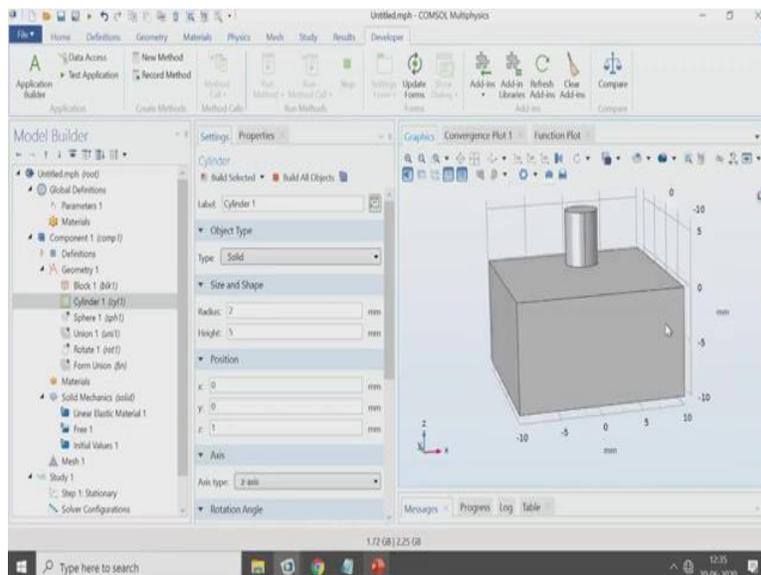
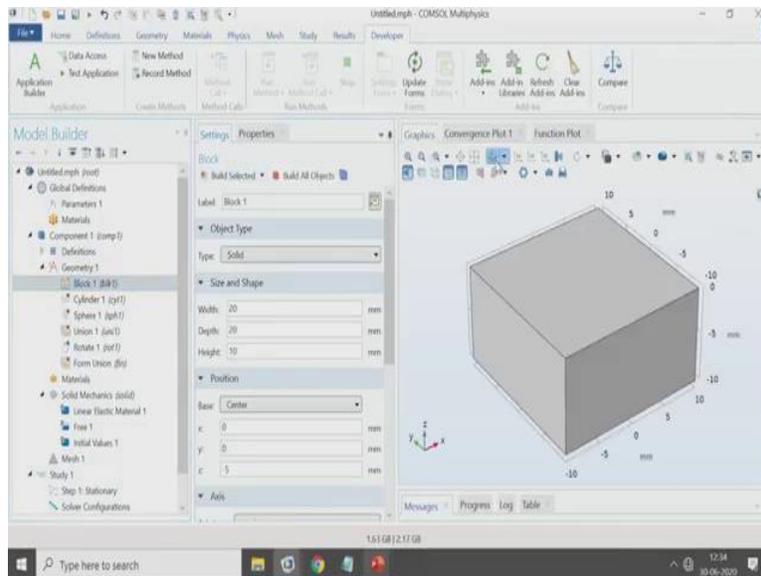
So this is the first thing when you see COMSOL. If you are new to COMSOL, it is always good to go with the model wizard. So, I will go to model wizard and I go to 3D space dimension because my geometry is 3D so we will go for 3D. The next thing is to choose the interface. Right now what we are trying to understand is the stress profile within the tissue; in that case we go for structural mechanics, solid mechanics. So for each interface, you can see the description is given on the right side.

So we just see the select this solid mechanics, add solid mechanics and then go to study and then we have different types of studies that we can perform. In this case we will go for stationary study that means it is a steady-state results. So at time t equals infinity, what is the stress profile in the tissue? You can also perform a transient analysis, that is a pressure which is developing and then it is decreasing maybe a Gaussian profile using the time dependent study. So let us go for stationary.

(Refer Slide Time: 05:26)







So this is the first thing that you see once you complete the model wizard. On the left side is the model builder as you can see. So the process to make any model in COMSOL is to follow the model builder top to bottom.

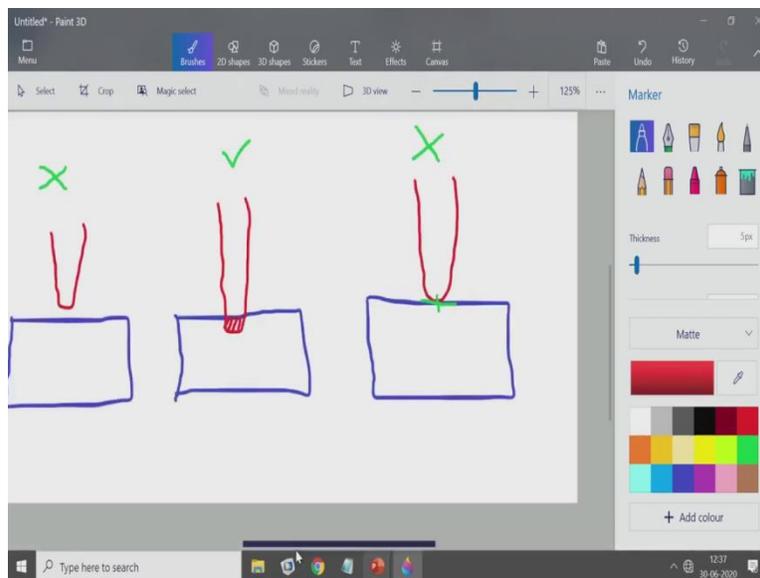
So first you make the geometry add the materials, apply appropriate physics domain and boundary condition, apply appropriate meshing conditions then apply which kind of studies you want to perform that is stationary or frequency domain study or time-dependent study and then finally post process the results. So your approach is top to bottom in the model builder and the alternate approach is to use the ribbon pane.

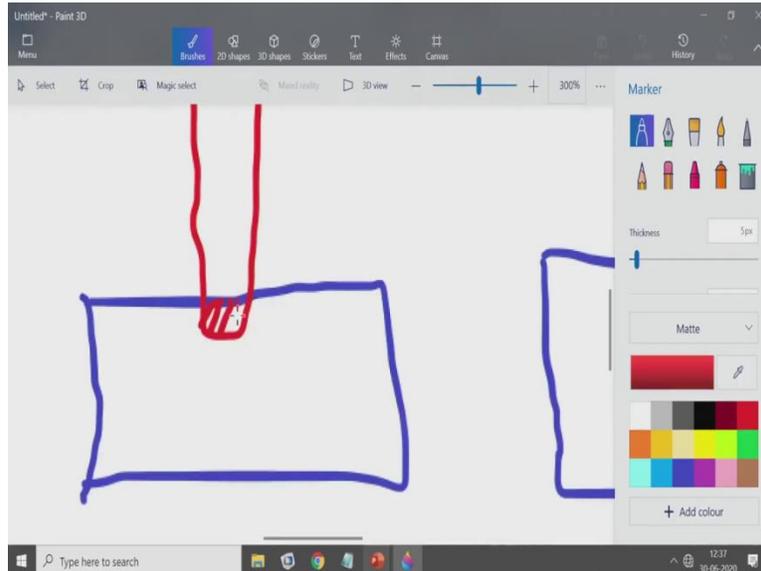
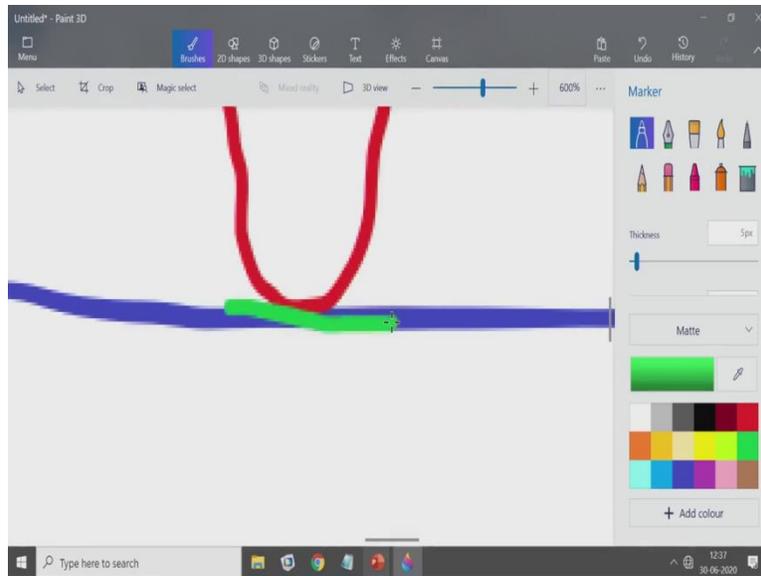
So, on the top you have geometry, materials, physics, mesh, study, and then finally post-process using the results ribbon pane. We also have a developer interface which helps you to write your own code or a script and then couple it with the model. So let us start to model this probe tissue analysis. So the first thing is to make the geometry, right.

To make the matter simple what I have done is I will import my geometry. So I will insert the sequence of the geometry. So you can see how we have made this the first thing is to make a block. So you can see that we have a block of around 20 by 20 and height of 10. So it is around 20 mm in length and breadth and the height is around 10 mm and we have move the block downwards by minus 5 mm.

So, that the corner is around minus 10 by minus, 10 by minus, 10 this point. The next thing to model the probe, we are using the cylinder. So I see you can see that we have used the cylinder and the cylinder is having some gap as of now from the bottom tissue. So this is my electrode and this is my block of tissue. Then to make curvature on the probe, I just add a spear.

(Refer Slide Time: 08:13)





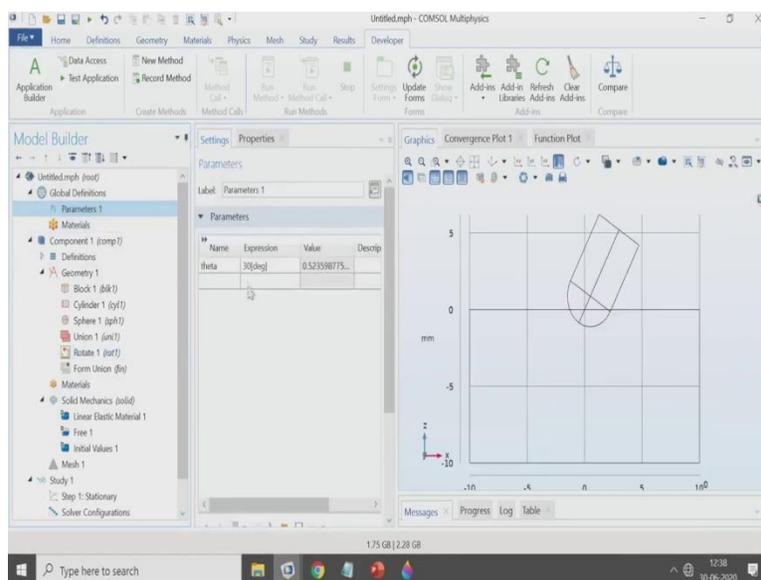
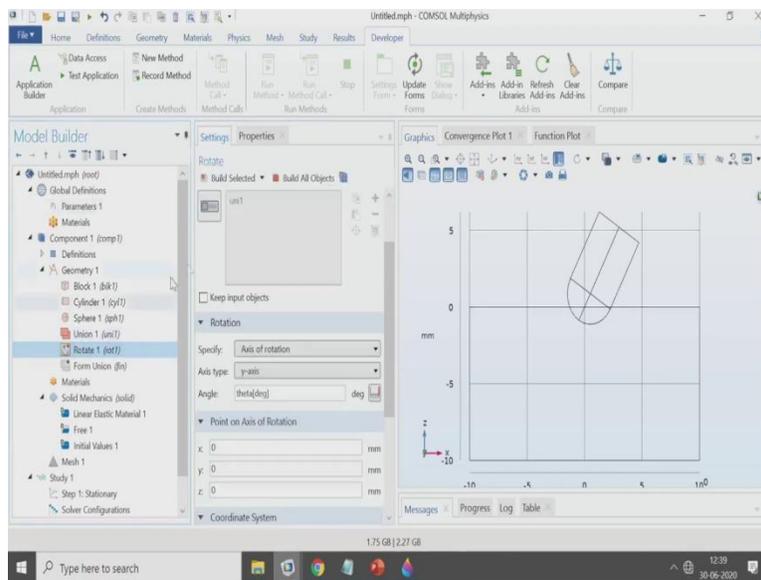
So the best way to model such kind of interaction is always to go with an overlap of domains. For example, there could be 3 ways to solve this problem; the first one is if you do a side view cross section your tissue is the bottom in blue colour and your electrode could be a little bit lying about your tissue that is one of the approach. The second approach lies when you have the tissue in the bottom and your probe is overlapping with the tissue.

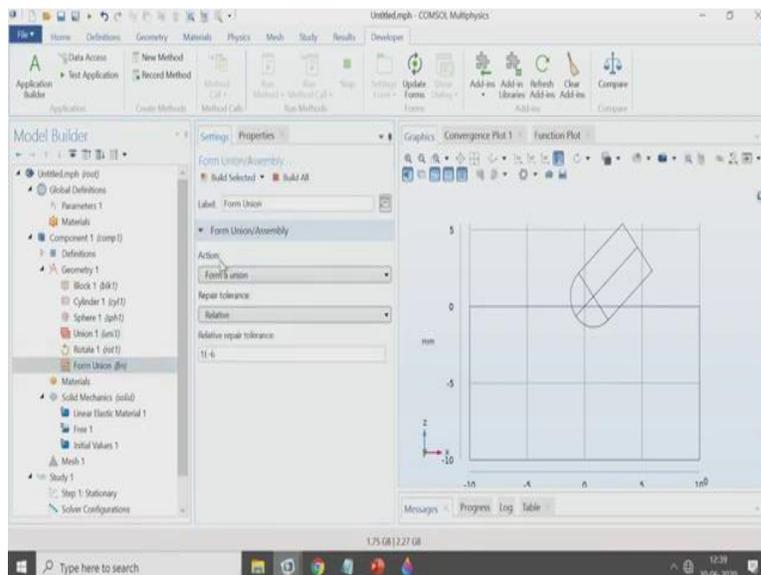
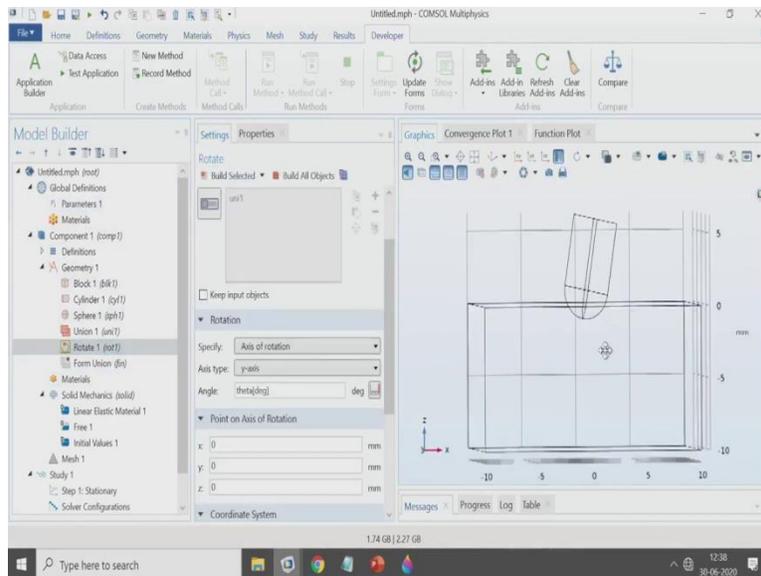
The third approach lies when you have the tissue in the bottom and the probe is exactly tangential to the tissue. So usually to model this kind of interaction, you cannot go with this because there is no interaction with the probe and the tissue. You can go with this, this is what

we are doing right now, the third option is that is numerically very challenging to model because of the tangential condition over here.

This will eventually be when you have at tangential contact, as people tell it, there is a point which interacts with the about probe electrode and the bottom tissue electrode and this point contact will lead to singularity issue. So we also do not go with this kind of approach. So what we do is we have some overlap of the electrode with the bottom tissue. This is why is; is we do in this particular model also.

(Refer Slide Time: 10:07)



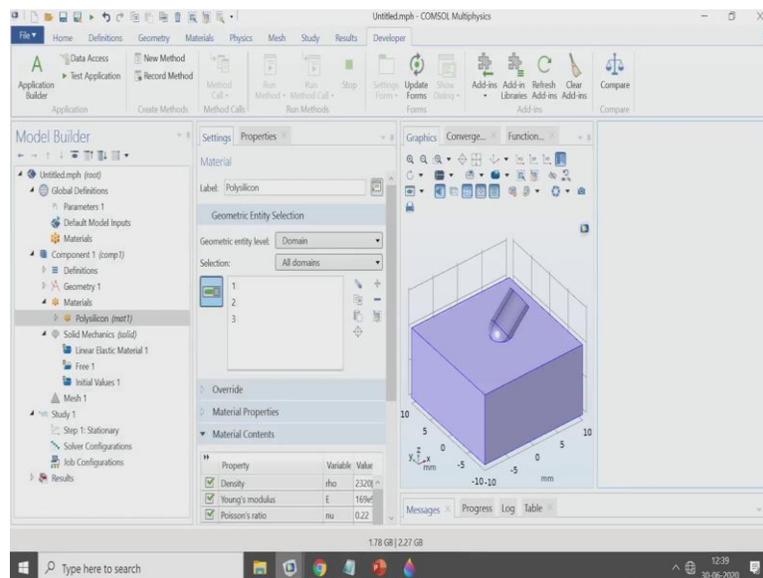
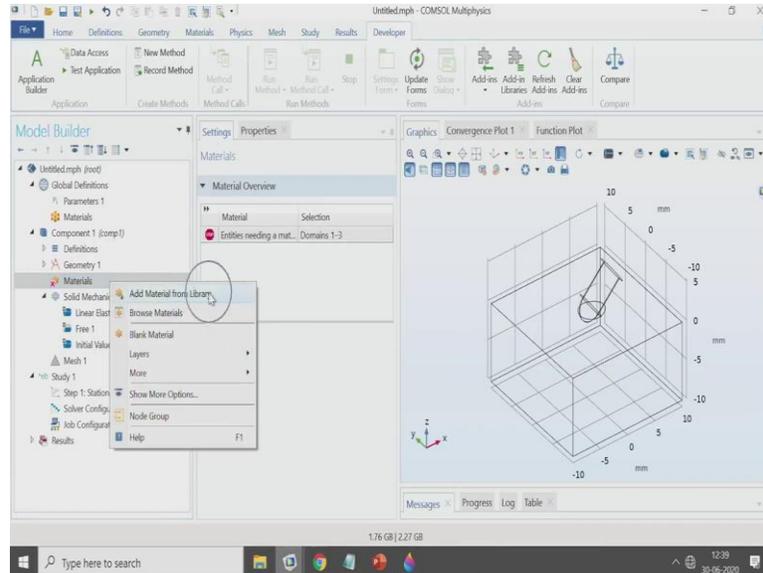


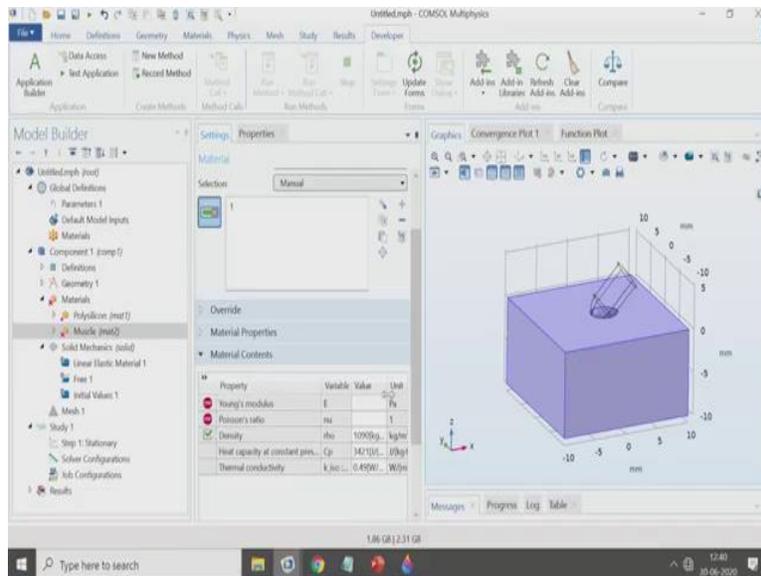
So you can see, if I just do a wireframe rendering, you can see if I do the XY plane. So you can see there is some overlap with the bottom tissue and the above electrode. Then I use the union to union my cylinder and sphere into a single object and then I use rotate feature. So this rotate feature is used to turn my or rotate my electrode at different-different angle theta.

So right now, if I put my theta as 0 degrees, I can see that my rotation it has zero, so it is not moving. If I make my theta 10 degree, you can see if I come to rotate, it is rotated by 10 degrees. If I go ahead, make it around 30 degrees, rotate, it is rotating even further. I can go ahead and make you 50 degrees for example, rotate. So this is how I am able to rotate my electrode and this

theta degree that is the parameter which I'm defining to rotate my electrode I will use this parameter in the parametric sweep.

(Refer Slide Time: 11:48)



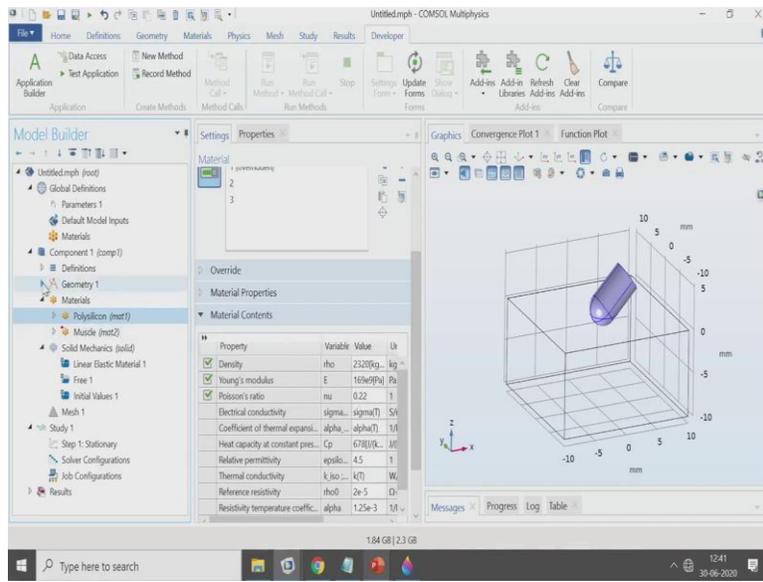
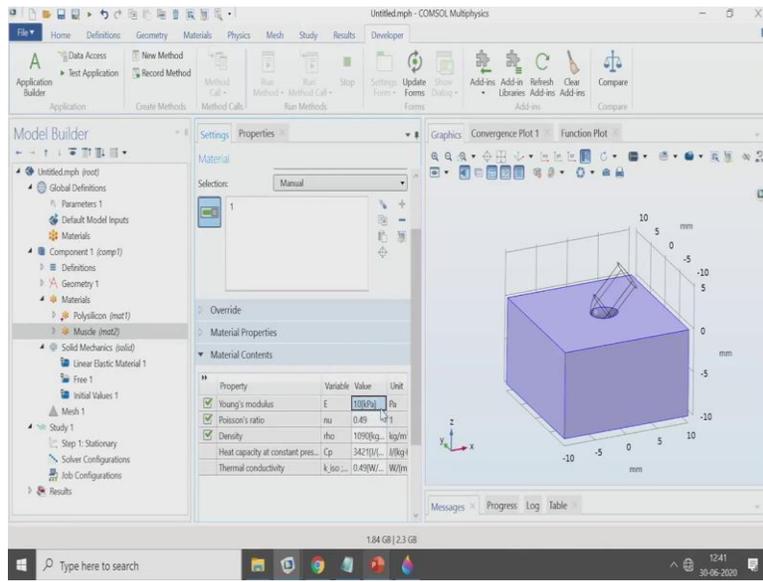


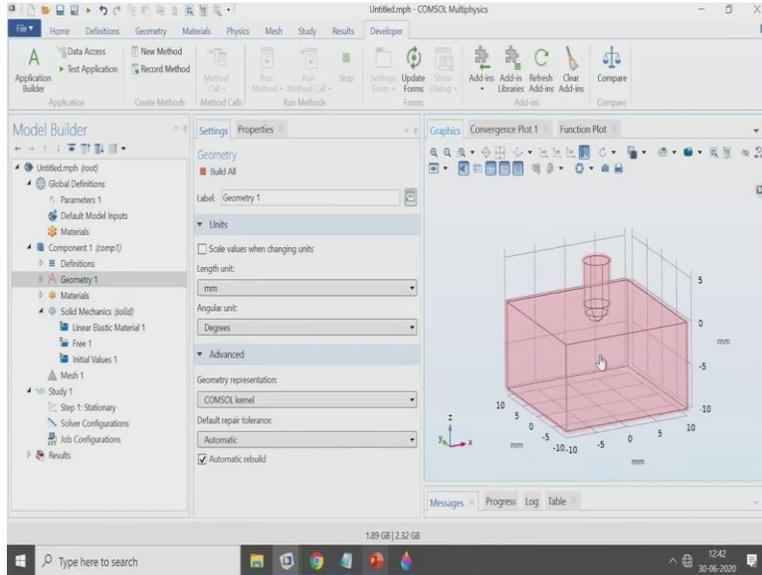
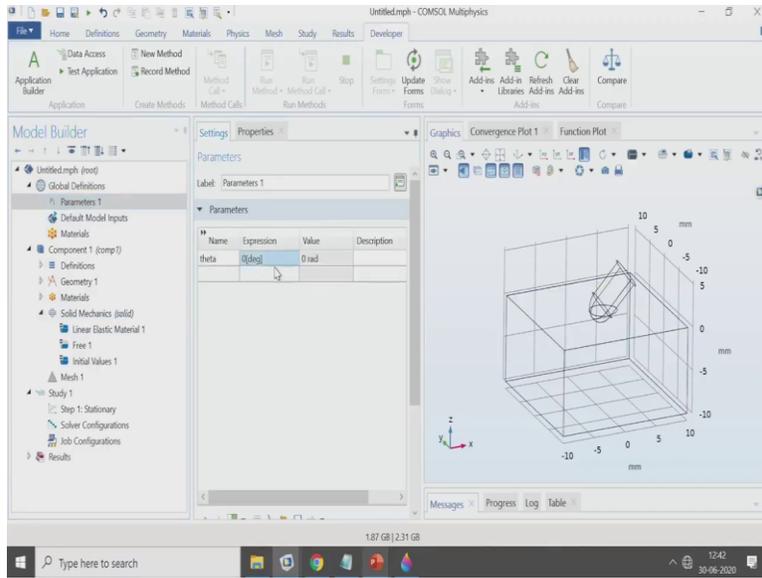
The next thing is to define the materials. So COMSOL has a huge list of materials. So if I just right-click on the materials, you can go to add material from library. So I go to built-in and I choose Poly-silicon. So by default all the domains are made up of Poly-silicon. So the first material that you add is assigned to all the materials that is by default. Now, the next thing is to add the tissue materials.

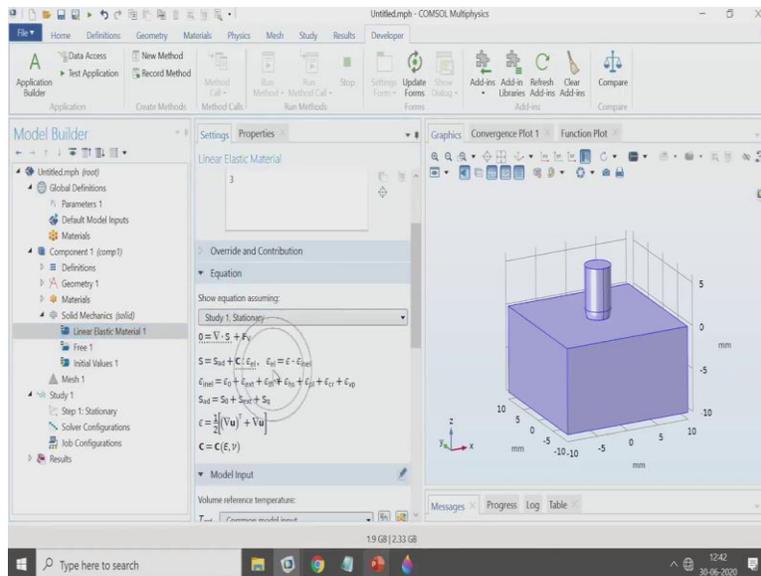
So we just right click on the materials, add material from the library if you go to bioheat, there are many different materials available. So these are basically the electrical and the thermal properties of the tissue. For example, if I just double click on the muscle, in that case, you can see that and I select this bottom domain. So I can see that the density is now available over here, the density that is tick mark, the green tick mark, means that the density model property is already available but you can see a stop button in front of Young's modulus and Poisson's ratio.

So this means that we need to enter this. So you can perform a literature survey from where you can actually find the Young's modulus and Poisson's ratio of the tissue and this is easily available.

(Refer Slide Time: 13:20)



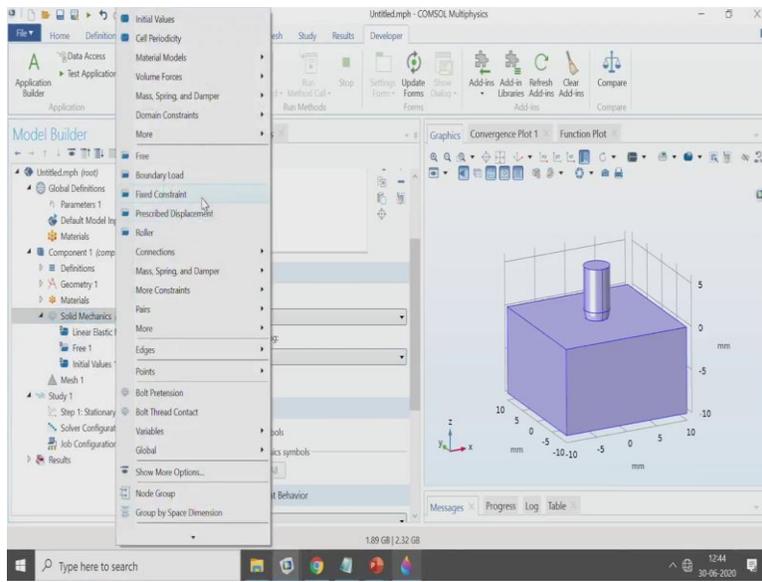
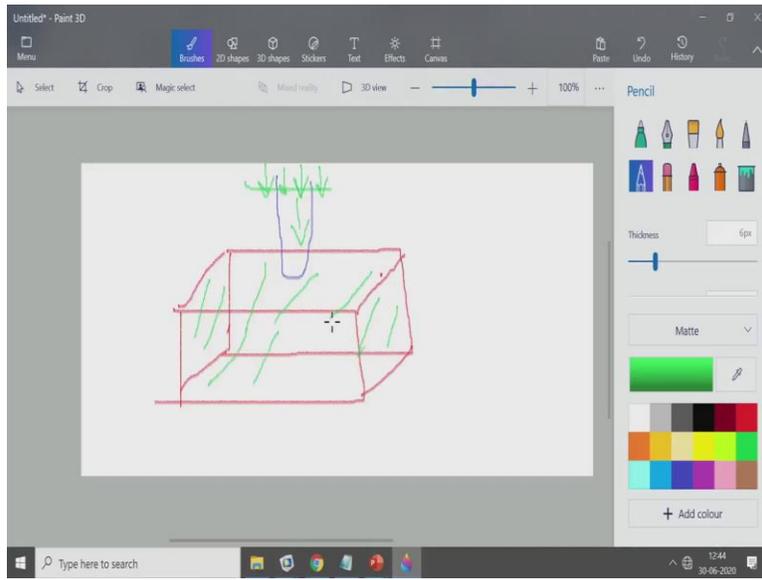


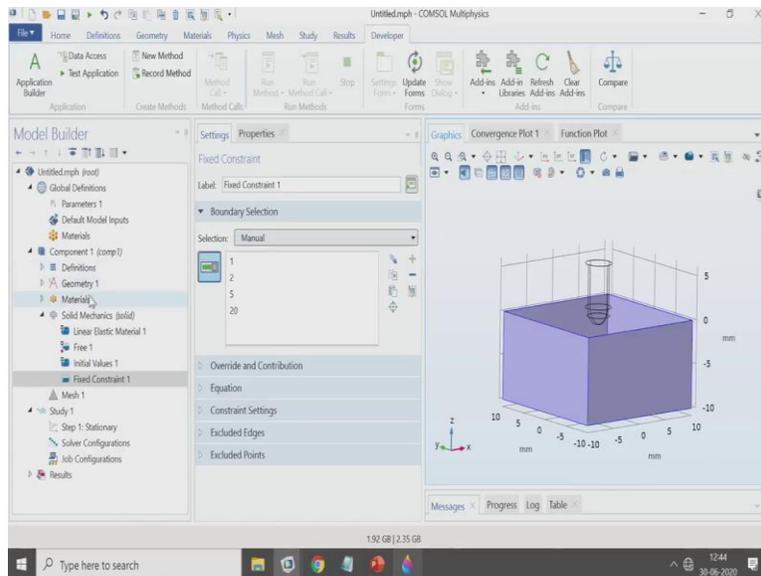


So I did some survey and I found that the, the Young's modulus of tissue comes around 10 kilo Pascal. So this is how I write this I write 10 in numerics and then in square brackets right kilo Pascals and then the Poisson's ratio I choose it as point four nine. So have Young's modulus and the Poisson's ratio and finally, the density which was assigned before and for Poly-silicon also we have already these three properties available; density, Young's modulus and Poisson's ratio.

The next thing is to apply appropriate physics boundary condition. For simplicity I will just go for 0 degrees, my angle and go to joy treat so to see it is normal. So the incidence of my probe is normal to that of the tissue. In solid mechanics you can also see the equation view which will help you to understand what are the equations being solved for and then you have a domain condition which talks about the material property of the tissue along with the equation that it is being solved for. So in the equation view, you can actually learn which equation is it is being solved for.

(Refer Slide Time: 15:03)



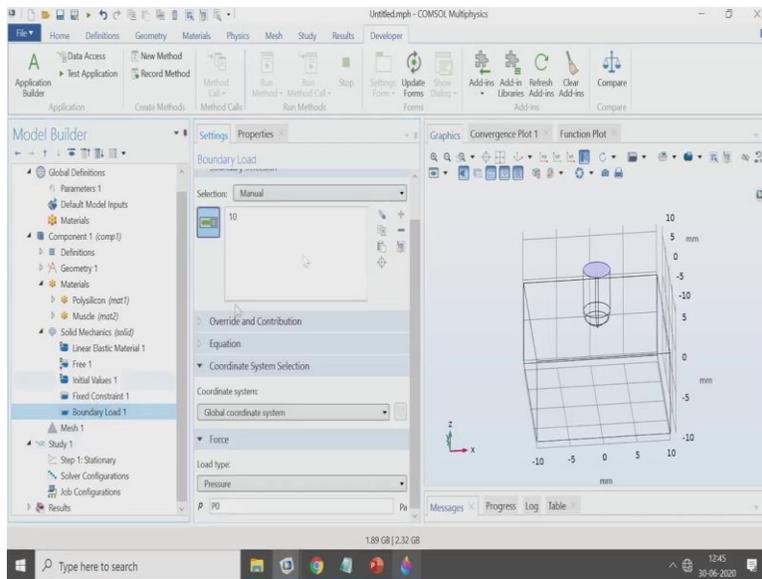
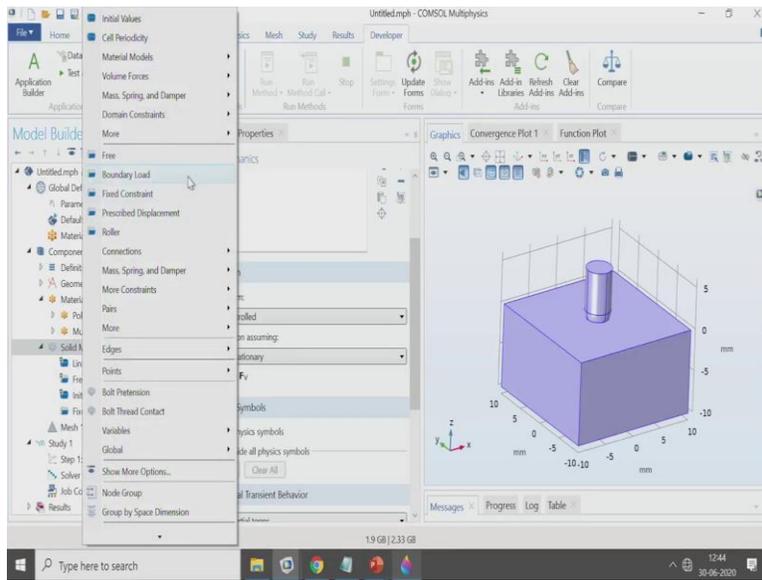


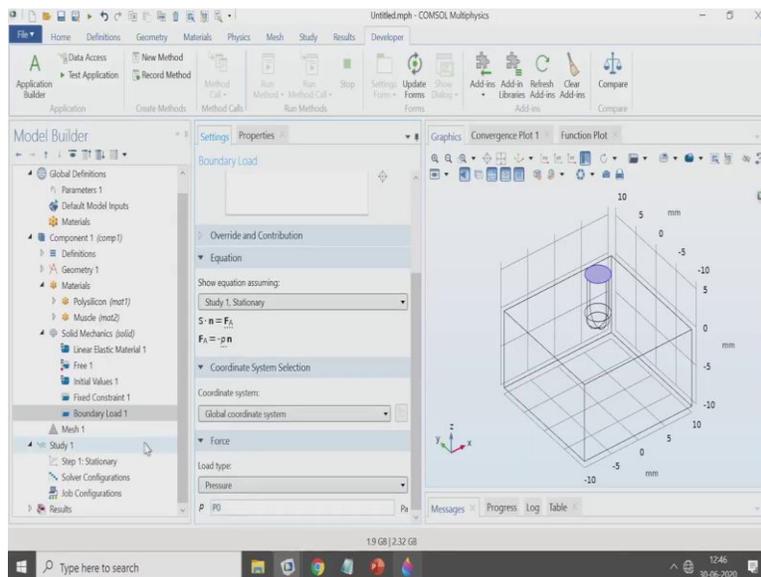
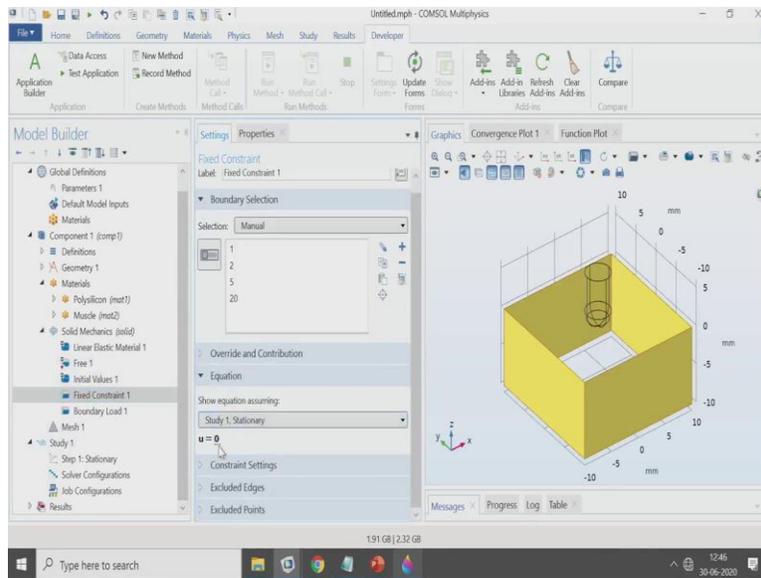
With solid mechanics what we want to do is if you have the probe like this, sorry you have the tissue like this and we applied the probe somewhere in the top like this. In solid mechanics, what we are going to do is we are going to apply the force in the top direction, so from the top direction towards the downward direction.

So we are going to apply the force to the tissue from the top to the downward direction. In addition to it, we also need to make sure that the tissue is not hanging. So in that case, we need to fix the tissue somewhere. So for example, you have your probe over here electrode probe over here see, this will be hinged to some tissue and this tissue will not be hanging somewhere, but it would be constrained somewhere. So it needs to be given some kind of a fixed constraint.

So let us see how to apply the fixed constraint. What we are going to do is, we are going to apply the fixed constraints on the walls, sidewalls of your probe tissue on the tissue. So how do I do that? I just right click on solid mechanics, and we have option of fixed constraint. So once I click on fixed constraint, I choose the site boundaries like this. So what does it mean is that the sides of the tissue are fixed, they cannot move in any direction that is what this means.

(Refer Slide Time: 16:53)



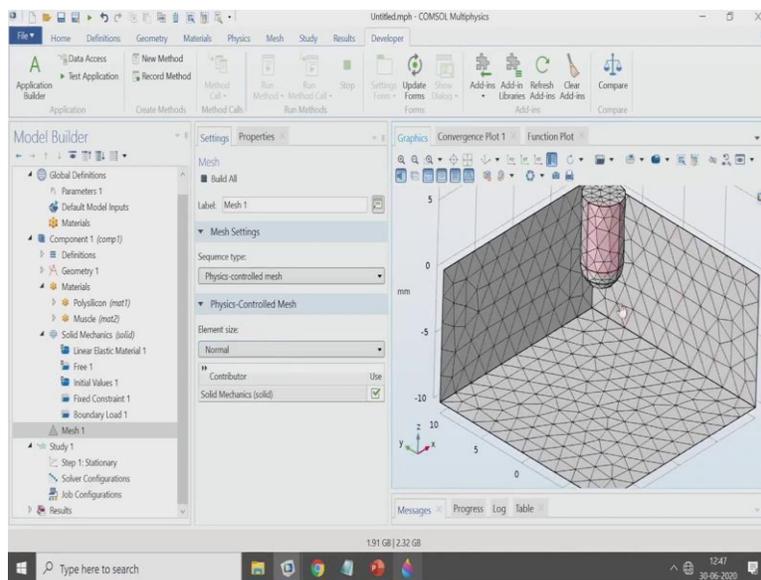
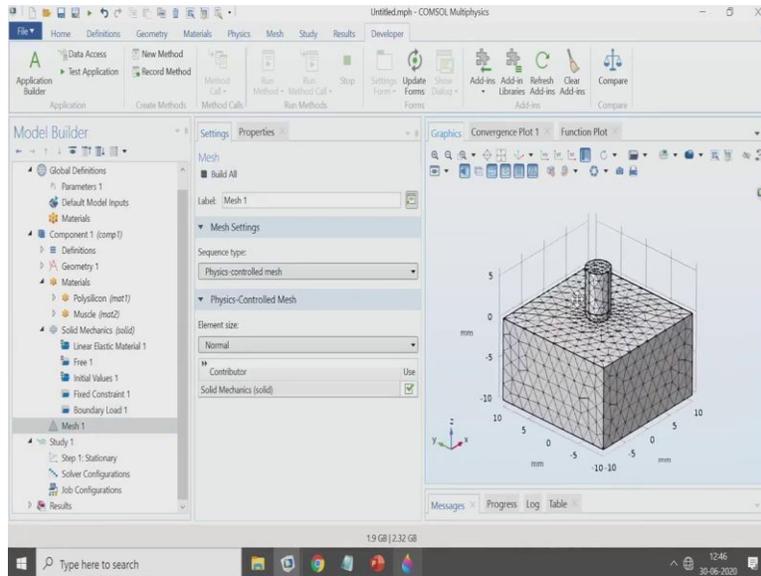


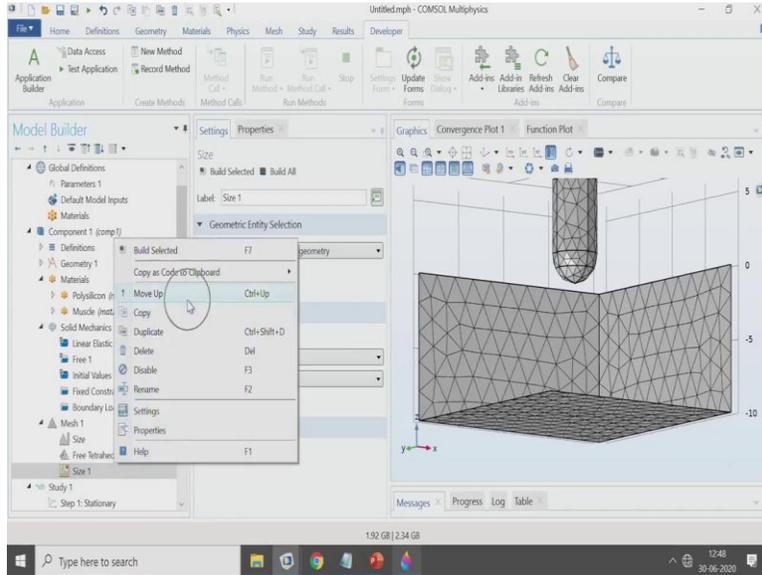
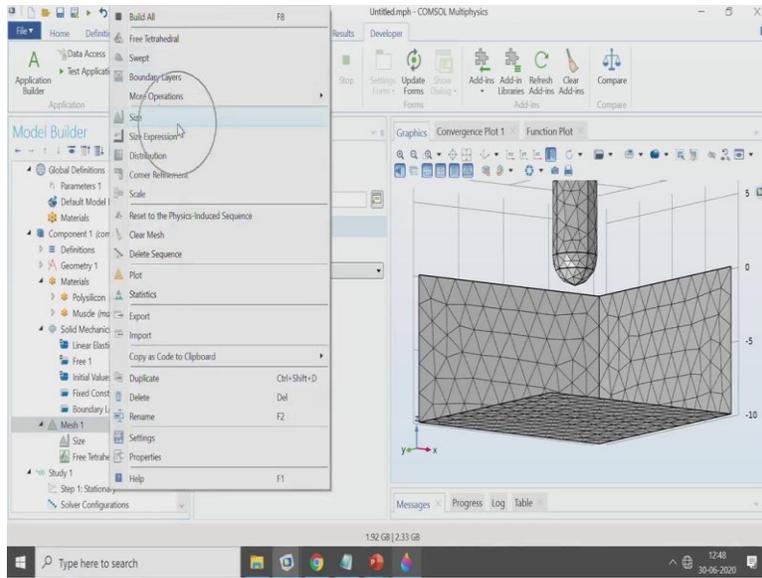
The next thing is to apply the force from the top of the electrode, how do I do that? I just right click on solid mechanics and apply boundary load. So, I choose my boundary load and I apply it on the top of my electrode like this and on the top, there are different ways to give the load. Either you can go give force per unit area, or you can give total force as of now what we are going to do is we are going to apply some pressure. So we apply around 200 Pascals of pressure for example.

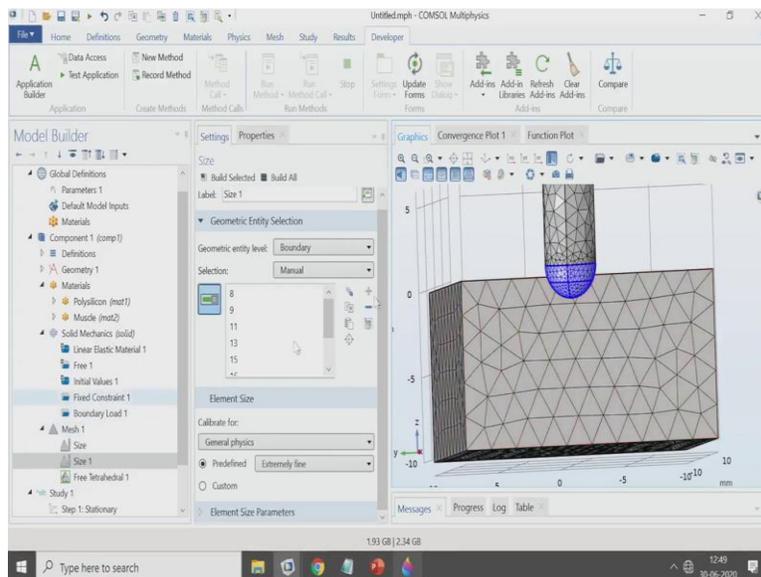
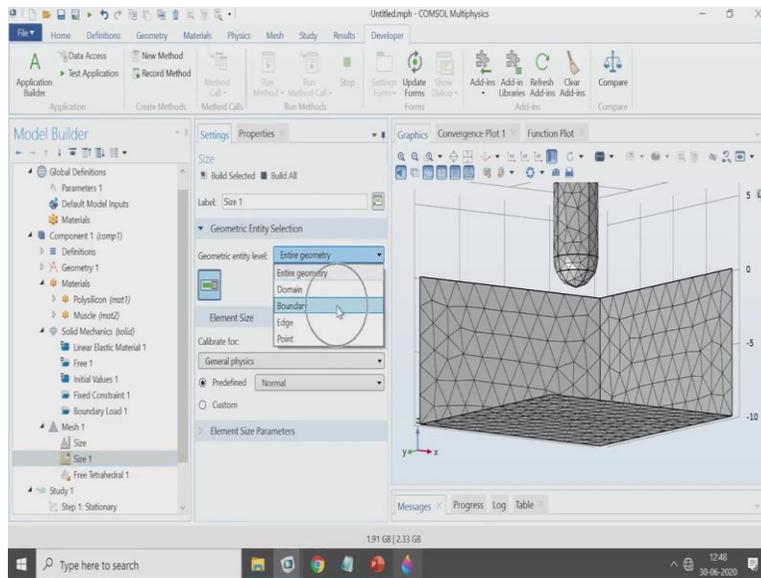
We can also use a parametric sweep. So I can write P_0 for example and this P_0 , I can define in the parameters and I can define it as 200 Pascals. This is my angle of indentation and this is applied pressure. So these are the only two boundary conditions that are required, the first

one is to apply fixed constraint and you can also see from the equation view that it means that the displacement on these boundaries are zero. We apply boundary load and from there you can see the Pascals that is F equal to Pa ; sorry P equal to F by A , those kind of equation and the applied pressure.

(Refer Slide Time: 18:38)







Now we go to the mesh and click on build all. So you can see the mesh profile as of now. But what I want to do is, I want to have a more refined mesh, at the interaction where there is interaction. So if you want to see within my geometry, I can use the click and hide approach. So I just use my click and hide, and I hide this particular boundary.

I hide also this boundary and maybe this boundary also, so that I can see the meshing that I am applying to my electrode at the interface. I again, click on click and hide, to disable my click and hide feature and now can see that with the normal Physics-controlled mesh, the meshing is not that great.

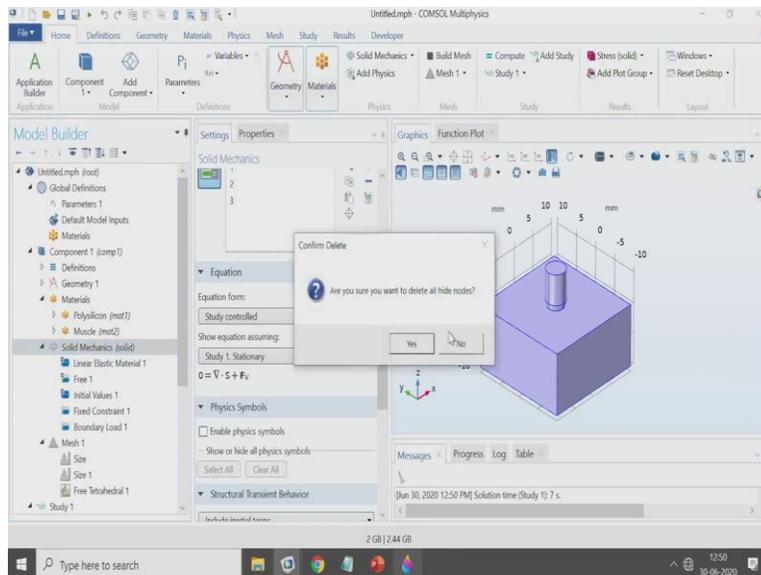
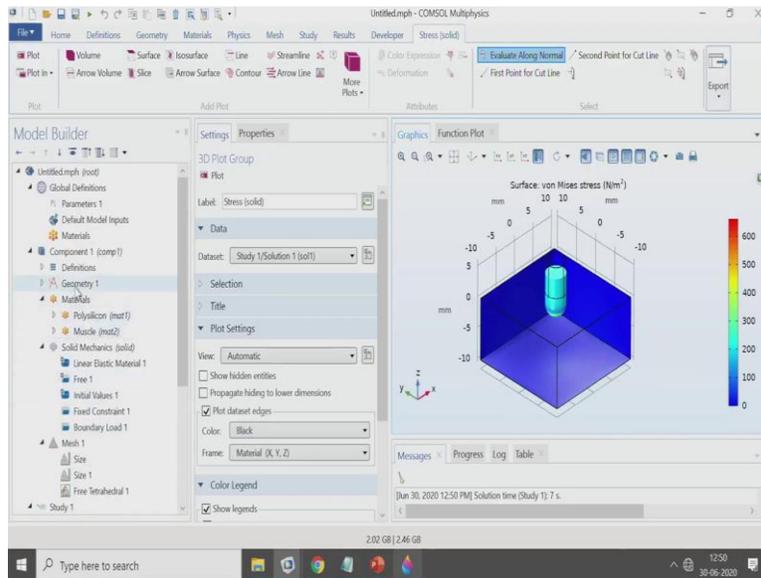
So either I can go to normal to fine, for example. In that case, also, it is not that great. So, but I, what I want is, I want to go with normal physics. But what I want to do is you can also go with this add study and compute your study, you will get some results. But what I want to do is, I want to make the (refine) the mesh, add the interaction to be more refined. How do I do that?

One way is to change from Physics-controlled mesh to user control mesh and over here I add one more size node to add the size node, just right click on mesh one, click on size and this size should be above your mesher. So this is a free tetrahedral mesher which meshes the complete geometry and these are the features which are given to the mesher.

So I need to right click on size one move up so now the size one will be also included while the mesher free tetrahedral meshes your geometry. In this size one instead of the geometric entity level entire geometry I use boundary. In that I choose the boundaries which are interacting with the tissue which are these boundaries and I perform a final mesh for these boundaries.

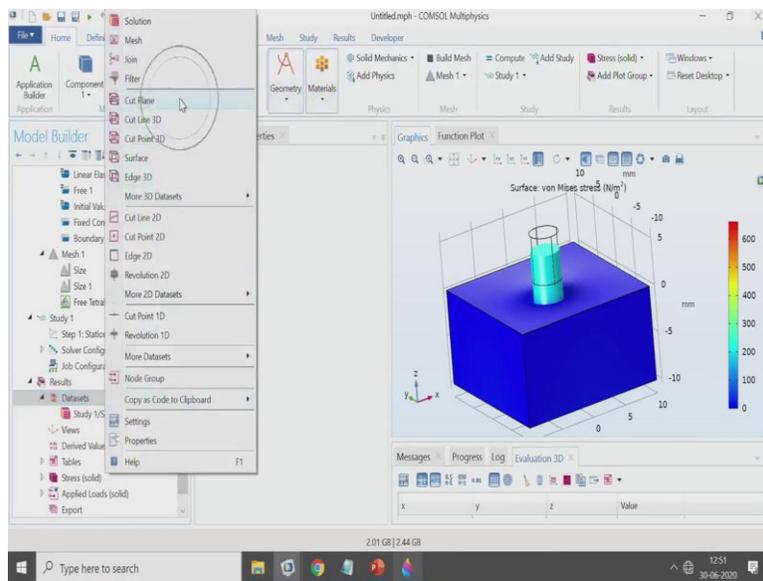
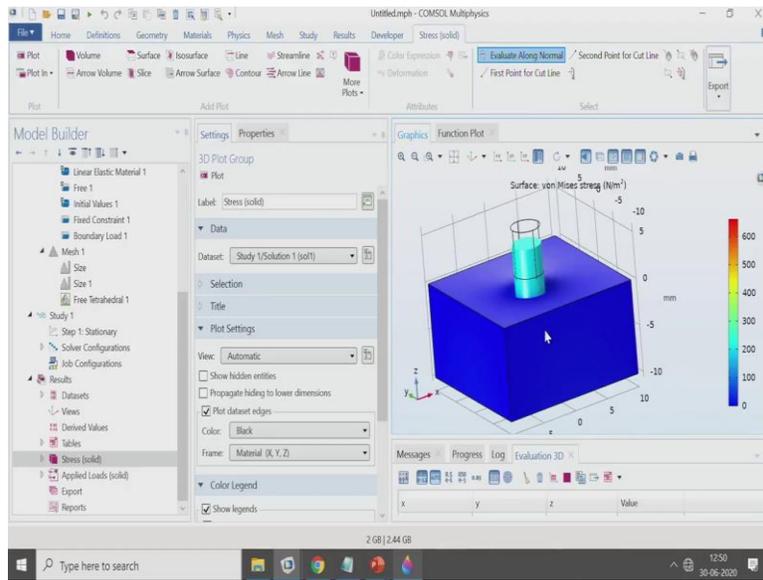
I can even go and make it extra fine. So what you can see is that only these boundaries are getting meshed with the refined approach. So not the other boundaries at the edges of your tissue, those are the same even if you go for extremely coarse. Right this is extremely coarse or extremely fine. So you can see that the outer boundaries of the tissue are remaining the same but at the interaction, the meshing has been refined and that is what we want because this is going to be the main place of our way to quantify the stress profile. So I just view all my domains as of now; looks good.

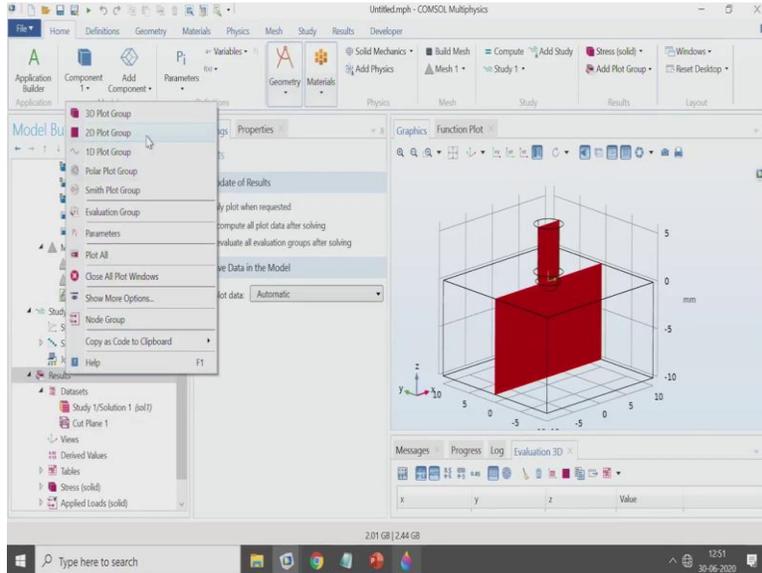
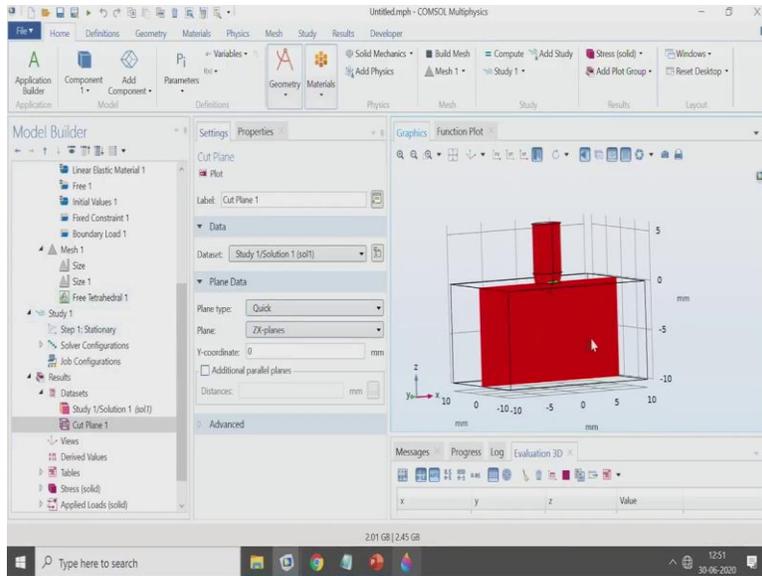
(Refer Slide Time: 22:03)

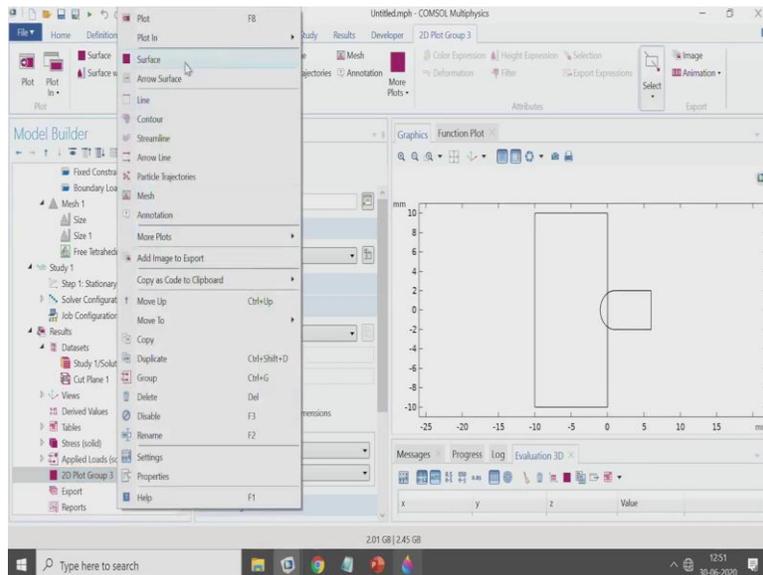
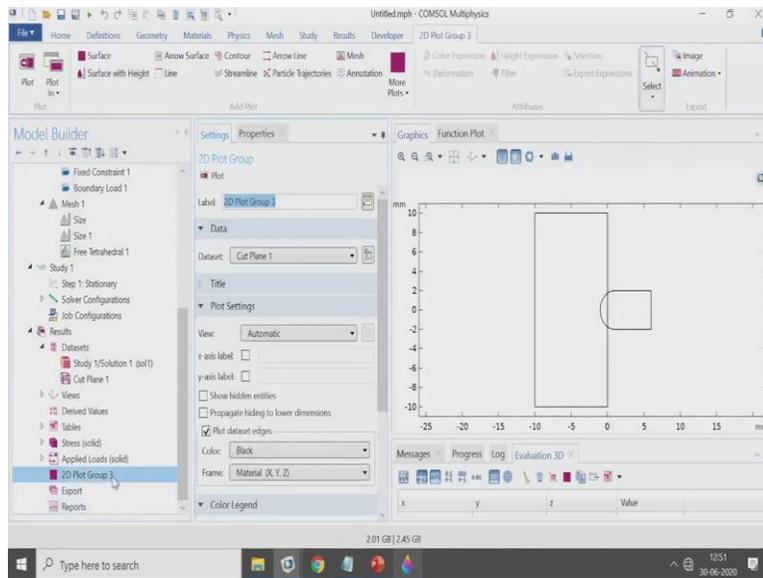


And then I go to my study and click on Compute and if my model was set up correctly, you will get some results. So, this is how the profile looks like right but this is not that, you cannot understand much from this figure. If you want to probe further from this figures, first thing is I will just try to disable, I will just delete all my hiding. So I just reset hiding, delete all hide.

(Refer Slide Time: 22:37)





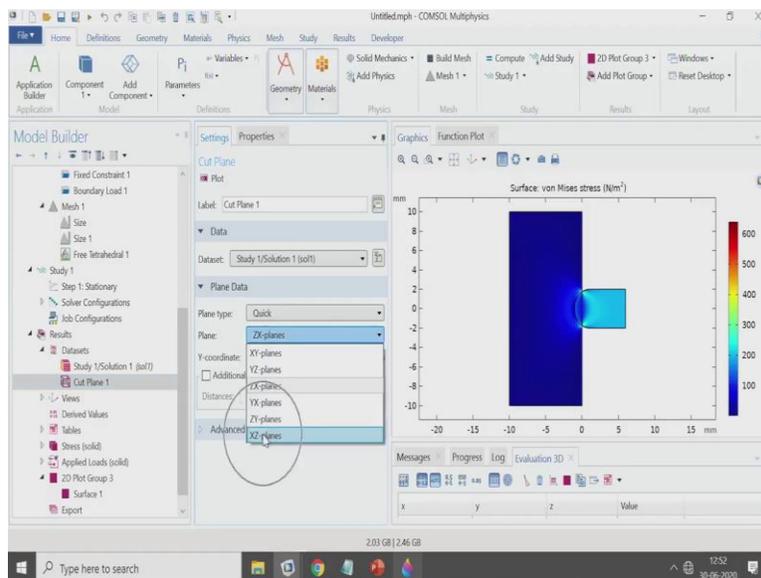
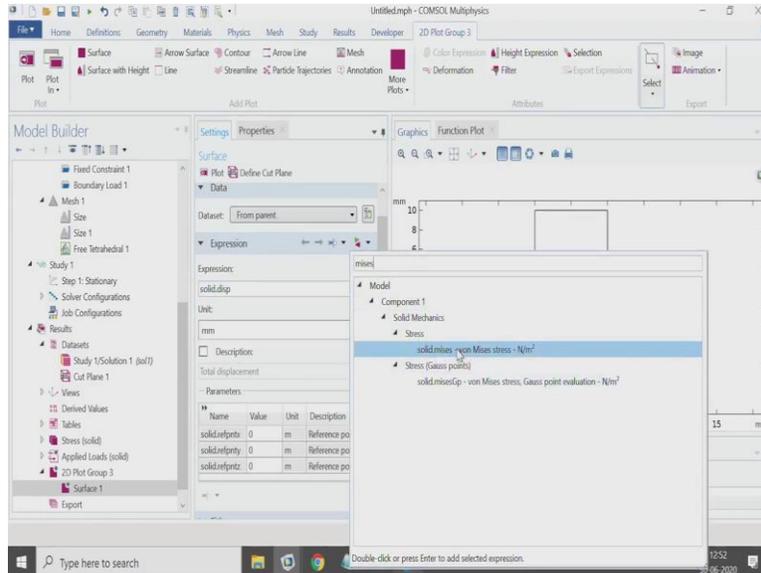


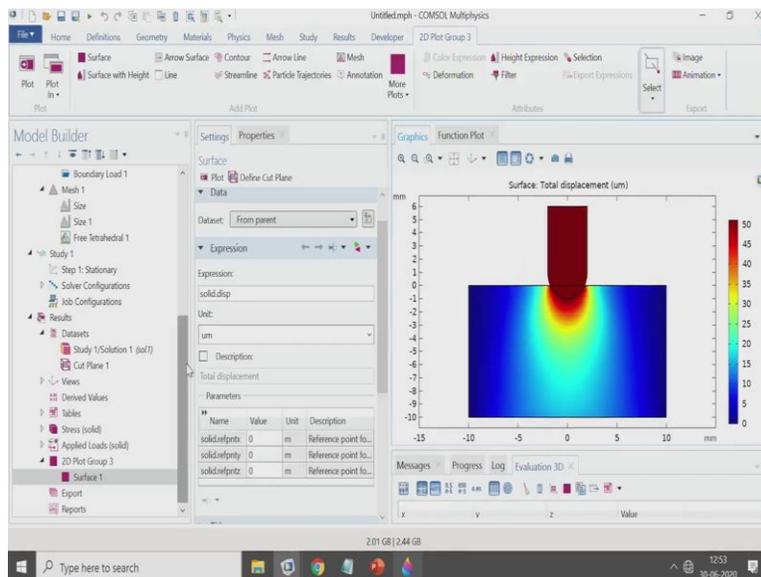
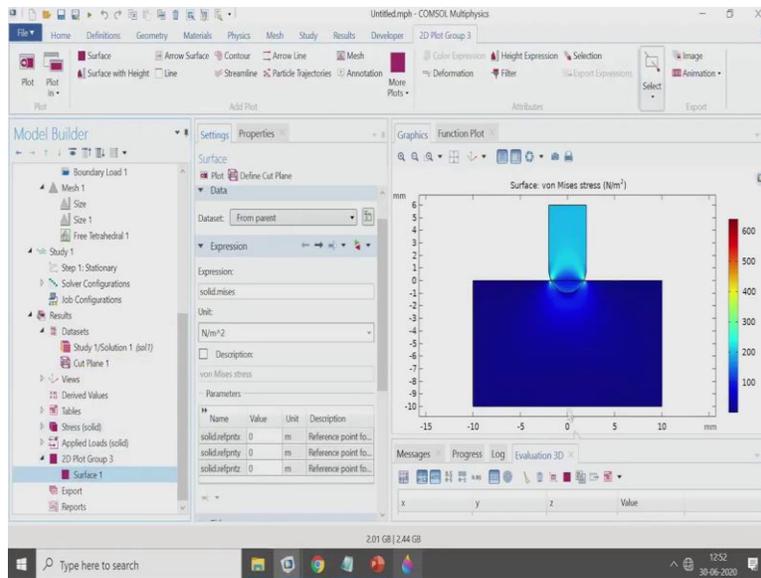
And now I can see from the top and you can see over here that there is some kind of a bulge inwards that is because the probe you can see this is the original position and then this is the final position. This probe is actually intending towards this tissue. But I want to actually evaluate the stress in a better way. One of the way is to do work cut slice through this domain and do the cut slice, you can use the data set feature, you just need to right click on data set and we have an option of cut plane.

So let us use cut plane and if we just click on plot, it is plotting the YZ axis but I want to do in Z axis because my angle of inclination is along Z axis. Now, I want to evaluate my stress profile at this cut plane that you see in the red colour. So I just right click on my results and I use a 2D plot

group because the plane is 2D. So I just used 2D plot group, in the 2D plot group, I right click on to the plot group and I click on surface plot. So by default, what you see is a solid dot disp.

(Refer Slide Time: 24:19)

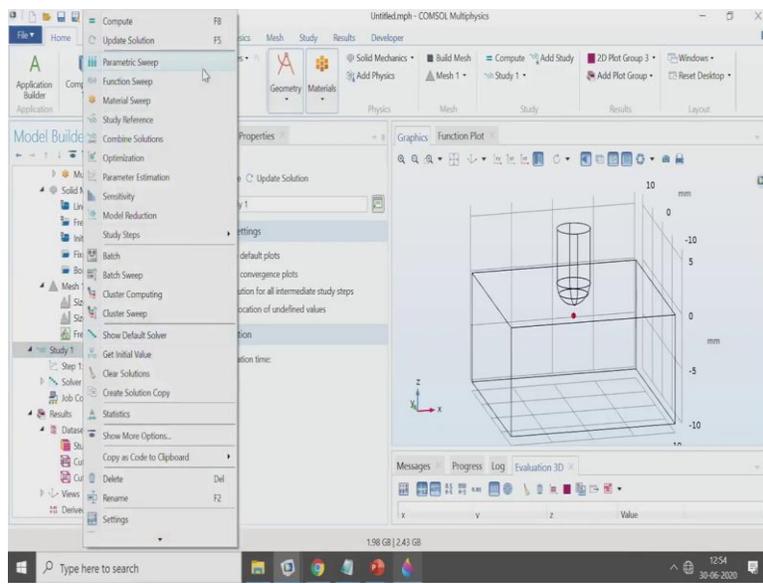
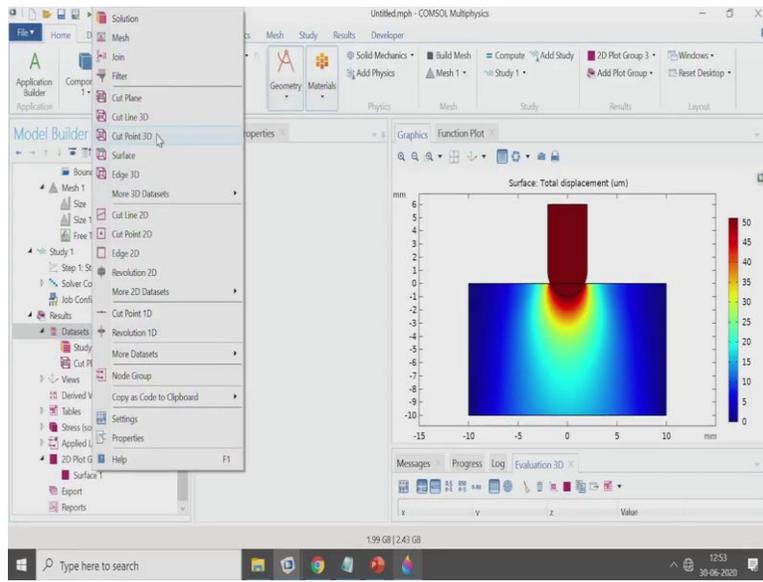


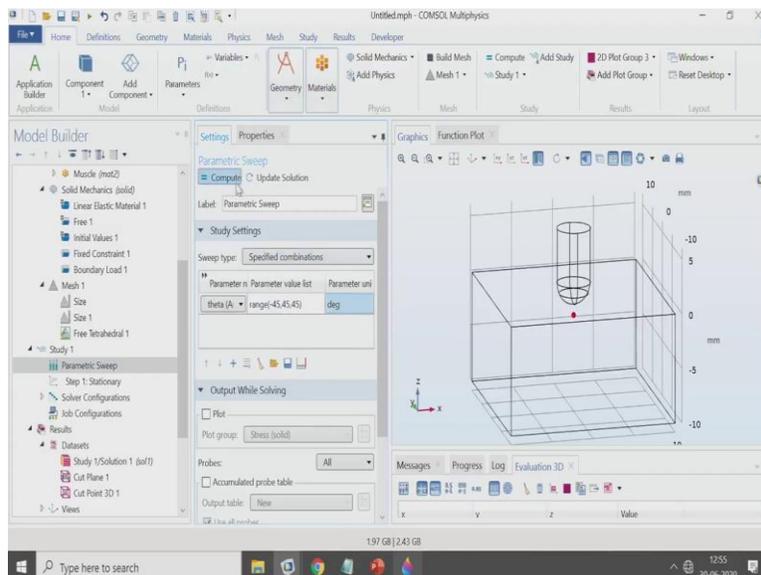
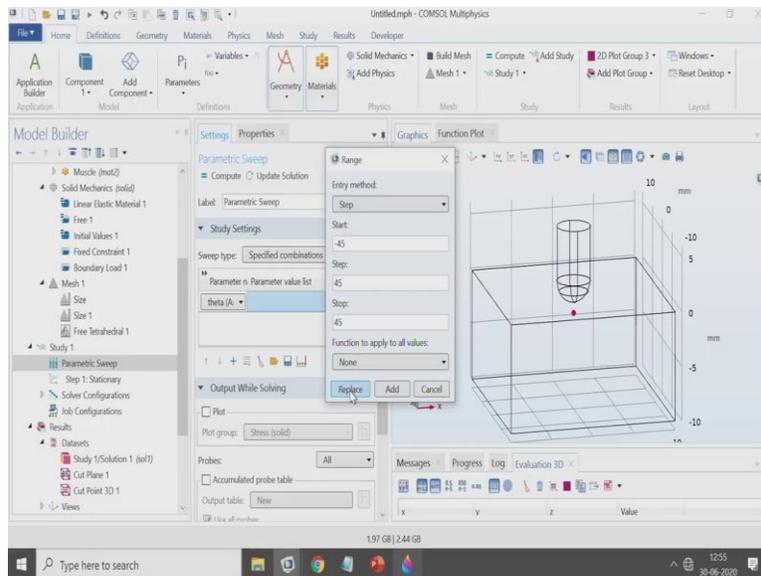


If you want to evaluate the stresses, you can just write mises and you will see the solid dot mises that talks about the stresses being developed on the tissue. So just double click on them and plot. What I see is the, my cut plane is like rotated by 90 degrees if you want to make it correct you can either use ZX plane, or XZ plane.

Now you can see that my cut plane is correct and if you see how the profile of the stress happens. You can also evaluate the displacement so instead of mises I can write solid dot disp and you can see how the displacement is happening. So if I do it in order of micrometers, you can see the red part is around 50 micrometers of displacement. So these points are getting around 50 micrometer displace downward in minus Z direction.

(Refer Slide Time: 25:46)





Now for example, I want to see with different-different angle of indentation, how the pressure at a particular point changes? So how do I do that? The first thing is to have the point where I want to evaluate, so to create a point, I just right click on the data set and I use mode 3D, sorry you can use the cut point 3D and in the cut point 3D, I use 000 to start with 000 somewhere over here, you can see your endpoint and that is 000.

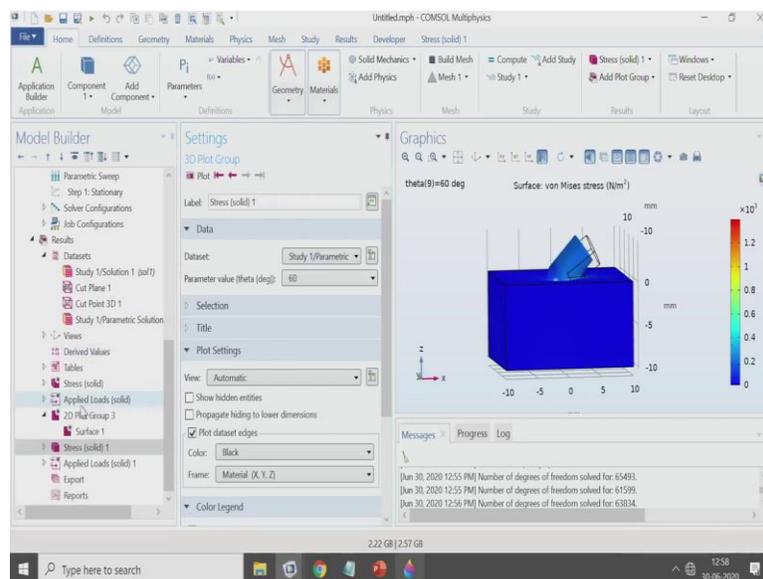
But I want to bring a little bit downwards, so used around minus 5 or minus 2. This looks good. So this point is somewhere below the electrode within the tissue and I want to see what is the stress being developed at this point? What is the displacement at this point? With a different-

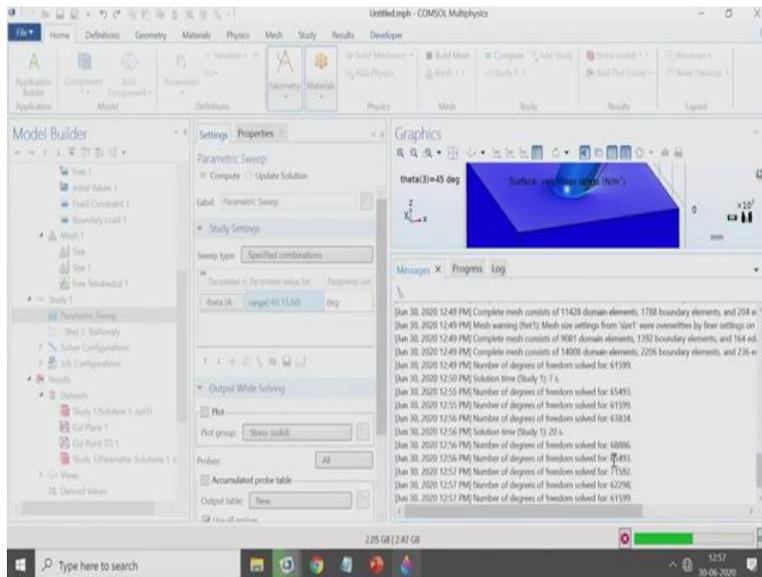
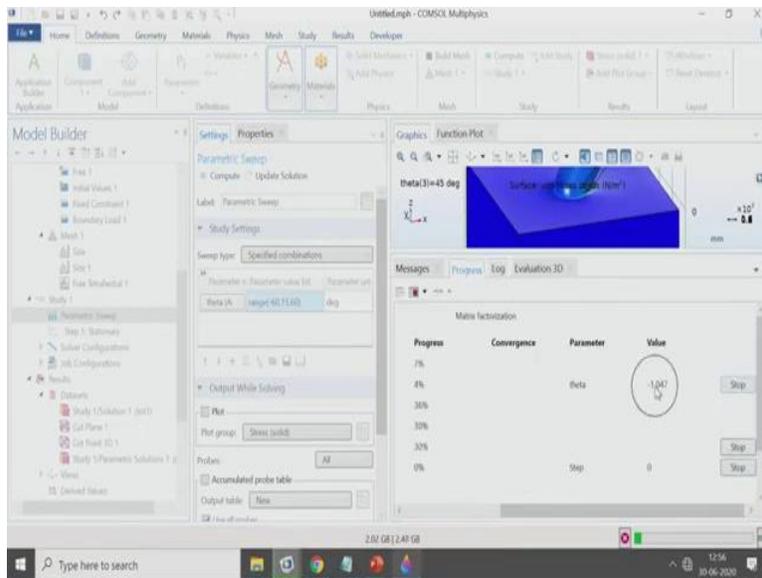
different angle of incident; to do this, I just right click so I want to do a parametric sweep. To do this parametric sweep, I just right click on study 1 and use a Parametric Sweep feature.

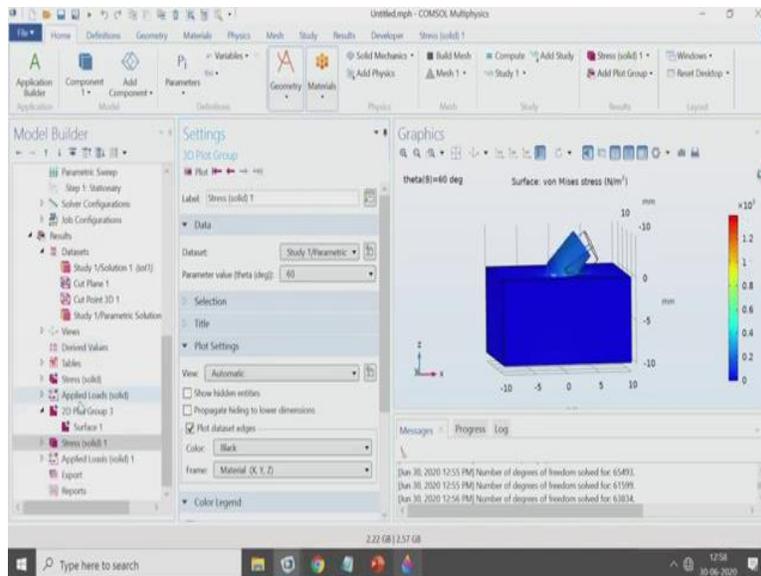
So let us use a parametric sweep and over here you can see an option of theta that is angle of incidence indentation and over here I can write like a range of minus 45 to 45 or you can also use the range feature. So I want to use the range feature for example. So I start with minus 45 degrees, step of 45 degrees and stop at 45 degrees, that means there would be 3 values of theta. The first is minus 45 degrees, second is minus 45 plus 45 because the step is 45 and finally the stop is plus 45.

So we do have minus 45, zero and plus 45 value of theta for the evaluation of stress at that red point. I just click on replace, and my units of theta is degrees. So that is it, I have added this parameter, and then I click on compute.

(Refer Slide Time: 28:08)





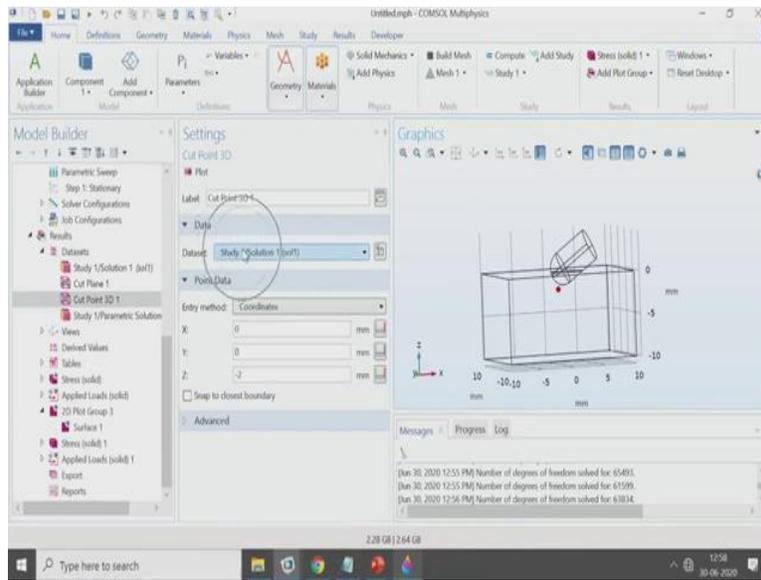


So here you can see for different-different angles, this is minus 45 degrees, then you have zero degree and then you have minus 45, plus 45 degrees. You can also, once you have run it for a lower number of degrees; you can also increase your number of degrees by making a more detailed sweep over here. So now we can see that we start with minus 60 degrees, but the step of 15 will move till 60 degrees.

So, you can also see in the progress bar, what is the angle of theta in radians, you can also see the log file to see how it is progressing. And the message bar also you can see the number of degree of freedoms that it is being solved for. This is a measure of how much time it will take for your model to solve.

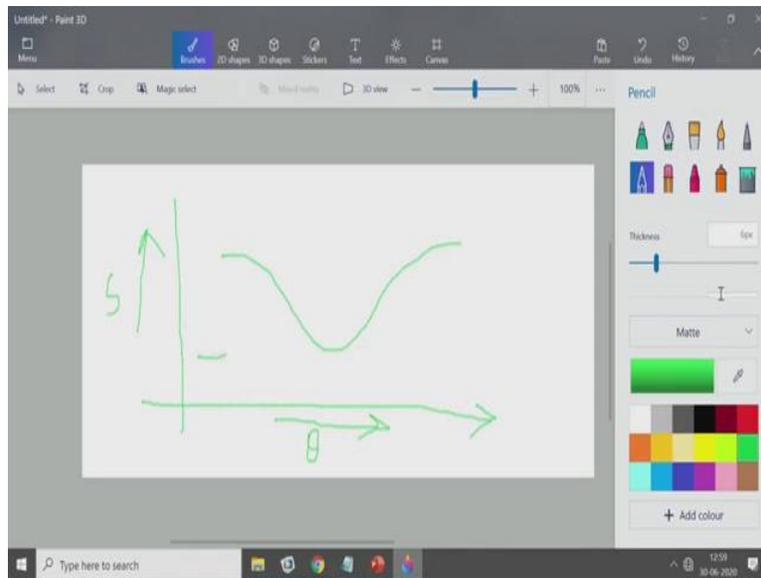
If it is in order of few 100s of 1000s, then it your model is not that big and it can solve fairly quickly as you can see in this model but it is, if your degree of freedoms are in order of millions of numbers, that is 1 million, 2 million, 3 million; in those cases, it might take a lot of time. So now we can see that we have minus 60 degrees, minus 50 by 45 30, 15, 0 degree 15, 30, 45 and 60 degrees.

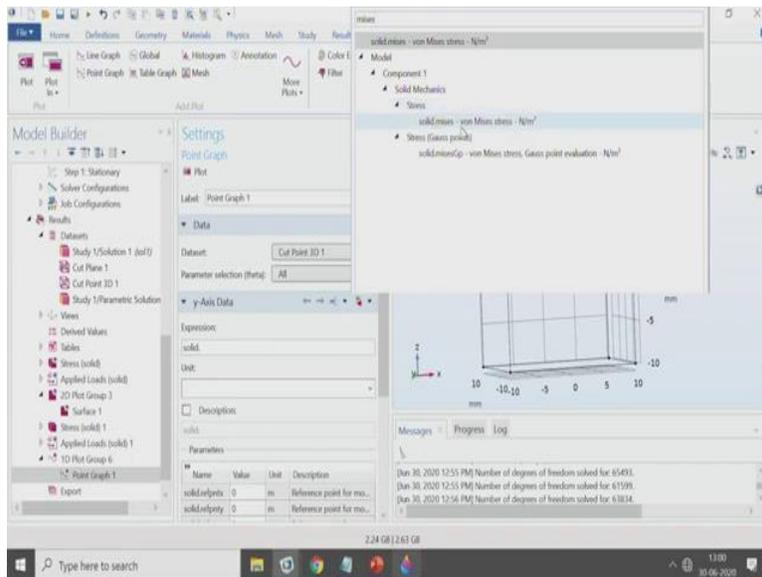
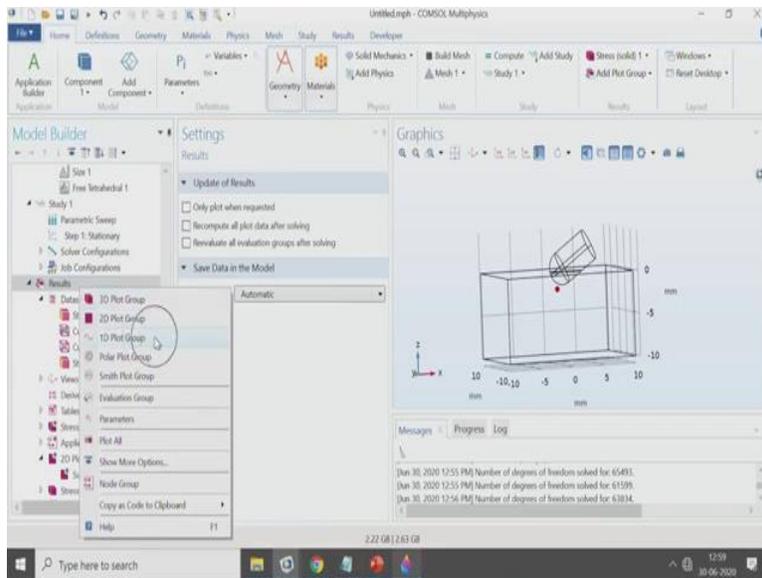
(Refer Slide Time: 30:15)

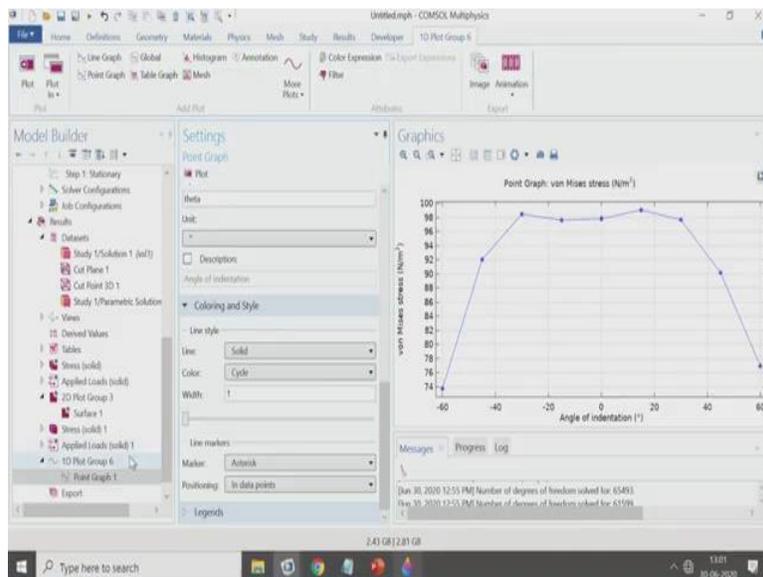
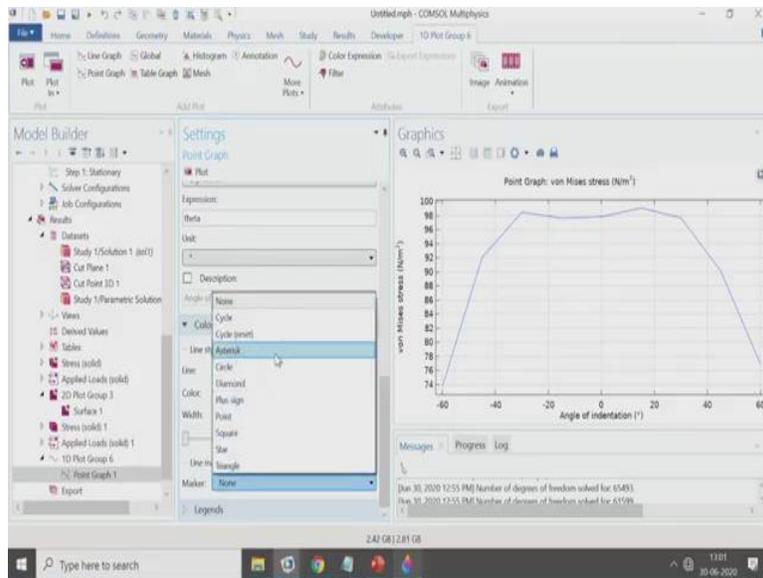


Now, what I want to do is I want to see the stress being developed in the point below, like the point over here for different-different value of indentation. In this case, my data set is set to study 1 solution 1, but I need to change it to study when parametric solution one, because study 1 is only for a single solution. Parametric study1 parametric solution is for the sweep of different parameters that we recently obtained.

(Refer Slide Time: 30:46)







Now, I want to have a figure something like this. My X-axis, I have the theta and my Y axis, I have want to model the mises, that is the stress in Newton per m square, for example and I want to see how this graph looks like. In that case, so this is our 1D plot group. So this is a 1D graph. So if you want to plot a 1D graph, you use the 1D plot group, so you just need to right click on the results.

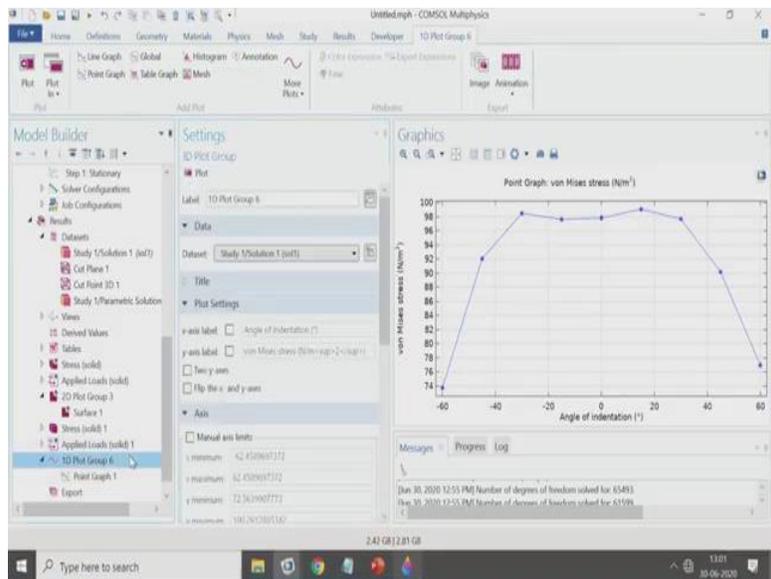
You have 1D plot group and over here, use a point graph because you are evaluating a point within the tissue. So I just right click on 1D plot group 6 and use a point graph. In this point graph, either I choose any of the points that I want to evaluate, as you can see, it is building adding up or I can choose from the data set itself, that is my cut point 3D one.

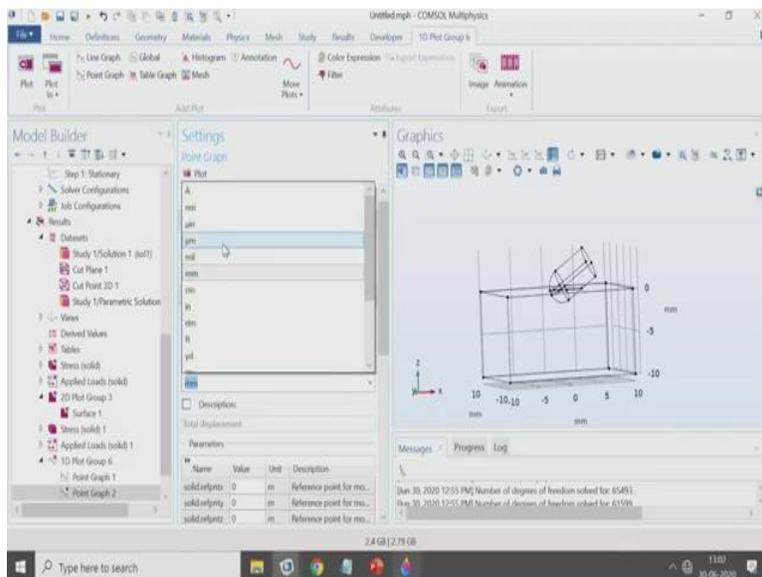
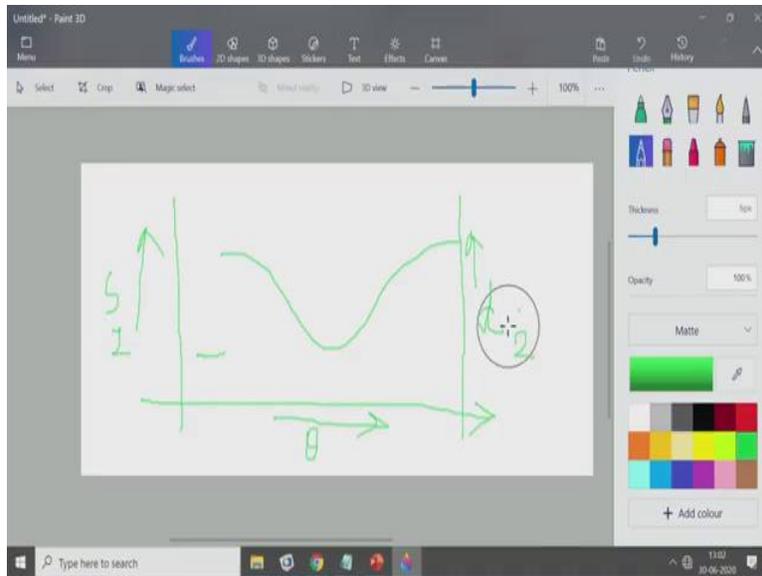
Now, what is what, what is it that I want to evaluate? I want to evaluate the stresses. So again, I can just go to expression, replace expression and I searched for stresses or just right mises. So you can see the 1mises stress, you can just double click on this and click on plot. Sonow you can see this graph of how these stress looks like as you change the angle of incidence indentation.

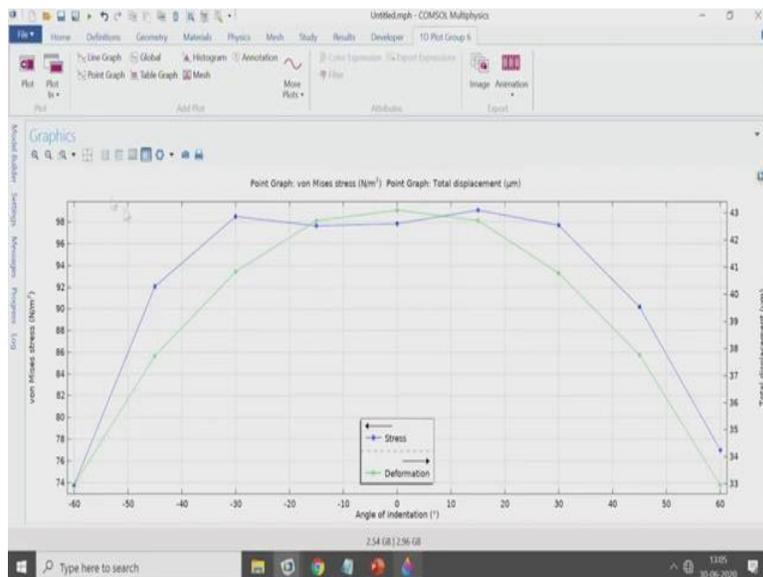
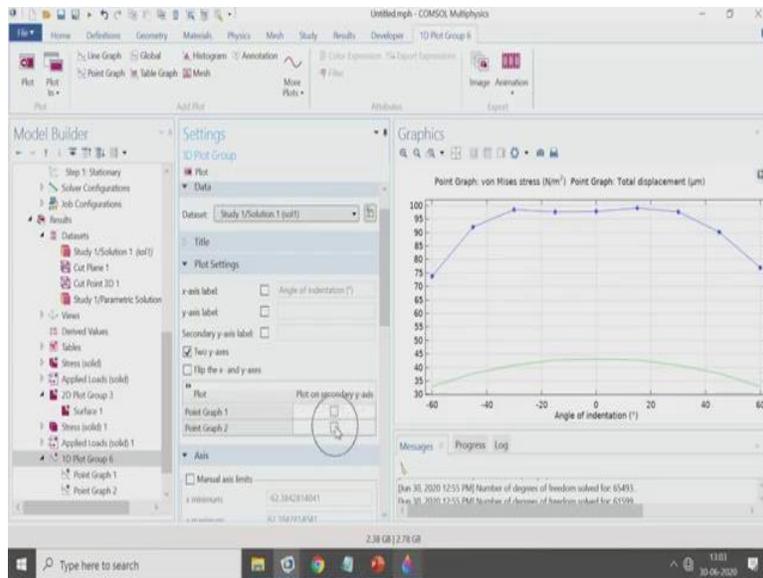
So I can also write it as expression and I write it as expression as theta and the unit as degree, so now, you can see that I have 0 degrees, you have 20 degrees, 40 degrees, 60 degrees on the X axis and if you go into coloring and style, you can use the line marker for example, to add a asterisk mark. So this will help you to understand at what particular parameter is the results evaluated.

So I just use the asterisk marks and instead of interpolated I can use in data point. So that means that at 60 degrees minus sixty degrees, the stresses were evaluated at 45 degrees the stress value and as you go on 30 degree, minus 15 degree,0 degree, positive 15 plus 30, plus 45 and plus 60 degrees at this points the Von Mises were evaluated. You can also make a more refined sweep to see a better smooth curve from this.

(Refer Slide Time: 33:57)







In addition to this, you can also plot the displacement. For example, I want to have 2Y axis plot. So I want to plot the displacement on the Y axis, second Y axis and my first Y axis I have stress in my second Y axis, I have the displacement. So I can add 1 more point graph. So I just right click on 1D plot group, use 1 more point graph and over here, I choose the data set cut point 2D 1 and over here by default, you can see you have the displacement solid dot disp. The variable which is required to evaluate displacement and the units I will mention it as micrometer and then I click on plot.

The issue with this plot is, first thing I need to also make it as X axis that is my, this X axis has to be the same expression as theta and degree but now I want to have my displacement on the Y

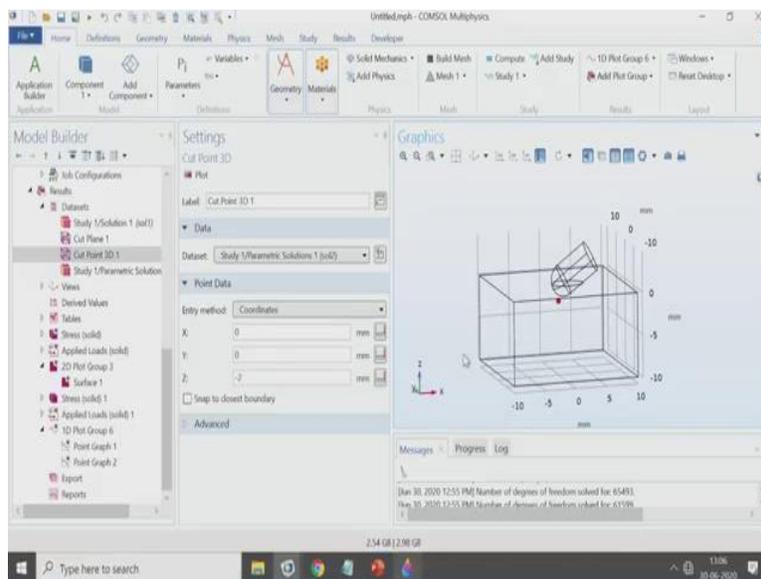
axis. So to do that, I can go to my 1D plot group 6, I can enable 2 Y axis plot and I can choose which graph to plot on the secondary Y axis. So I can use the point graph 2 for example.

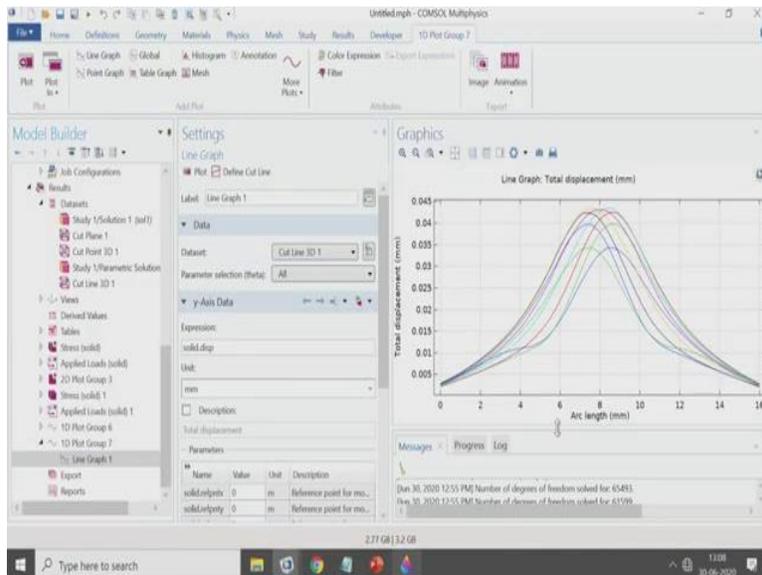
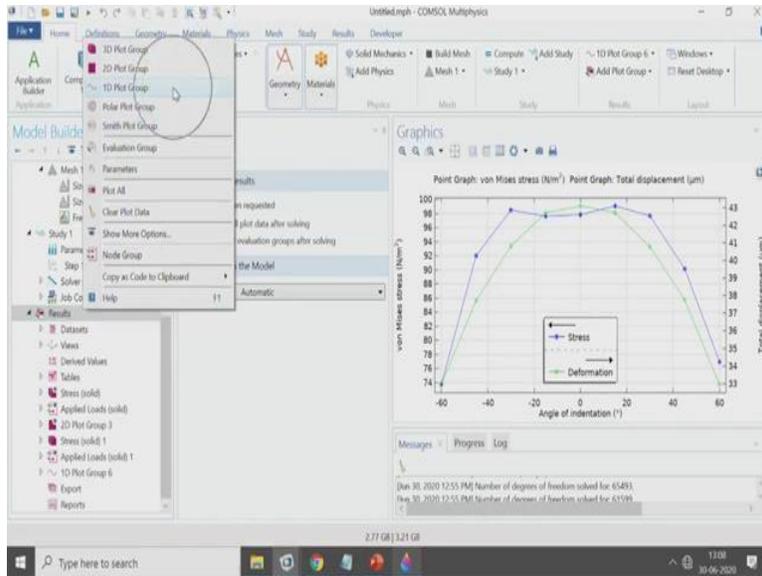
So now you can see that we have the displays sorry the, the Von Mises with the stress on the left, and the total displacement on the right. Now I want to have a legend, which tells you about which graph, which graph is which one. So in that case, I can use my Legends. First, I can go to my point graph 2 and add the markers. In this case, I use circle in data point for my displacement on the right side, and then I go to point graph 1, I go, my to my Legends, show Legends and I use manual.

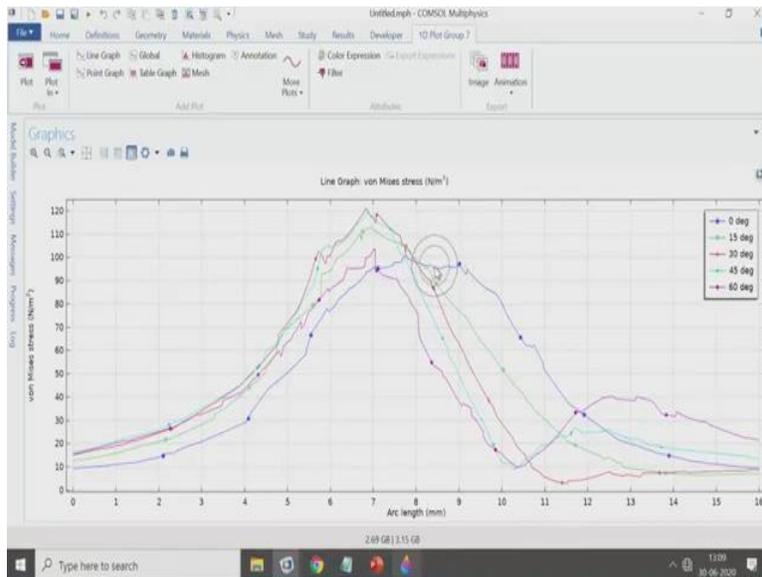
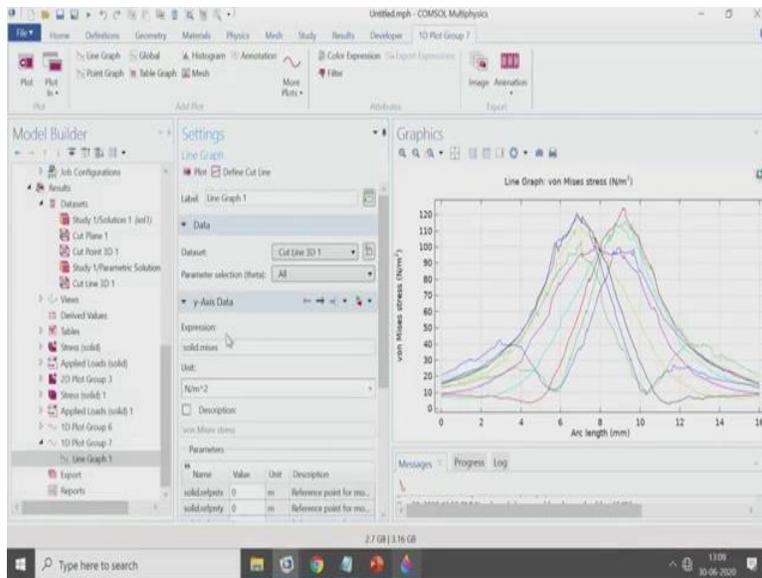
And I can enter over here, stress and I can go to my point graph 2, show Legend, manual and over here, I enter as displacement or deformation. I can also move my Legend somewhere, which is better for me. So I can just choose this option of position. So I use this position as middle, lower-middle, somewhere over here. So this is how my graph now looks like; looks good.

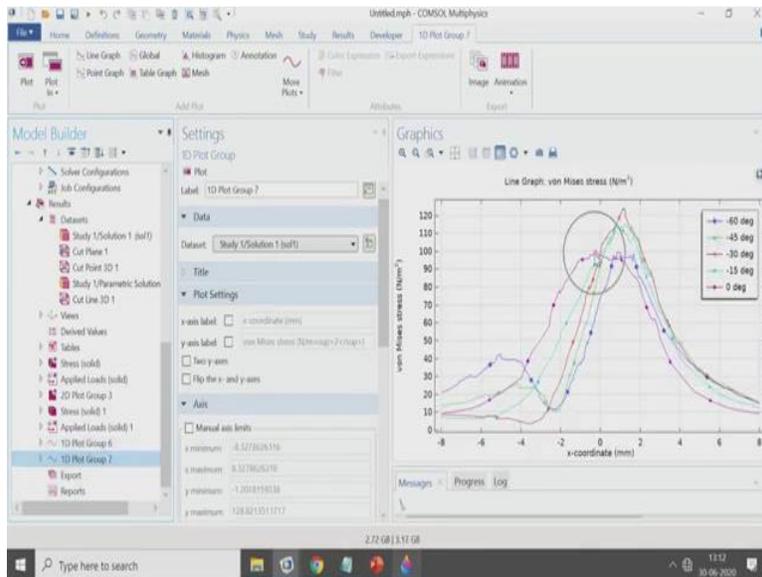
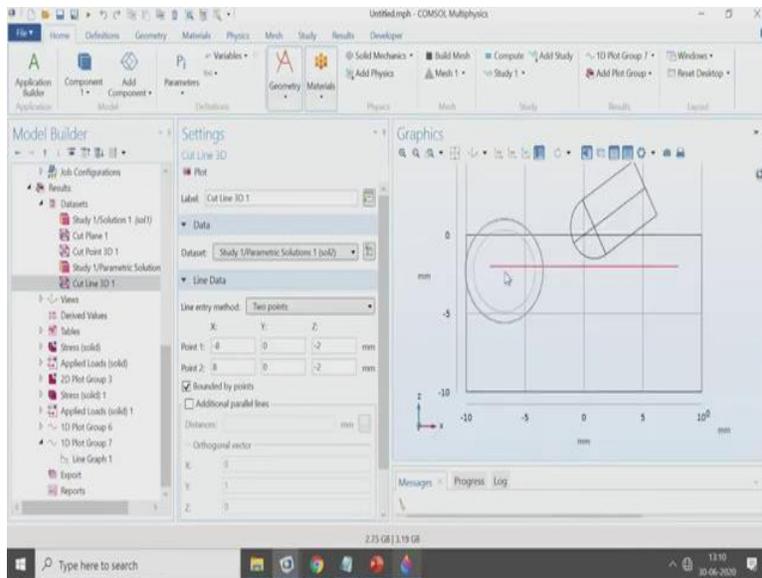
On the left, we have the Von Mises stress and on the right, we have the deformation, but total displacement and from this, you can also understand that the angle of indentation at 0 degree, the highest number of deformation and stress is being developed. But somewhere with around minus 30 degrees, it is even higher and then it reduces as you go on increasing your angle of indentation.

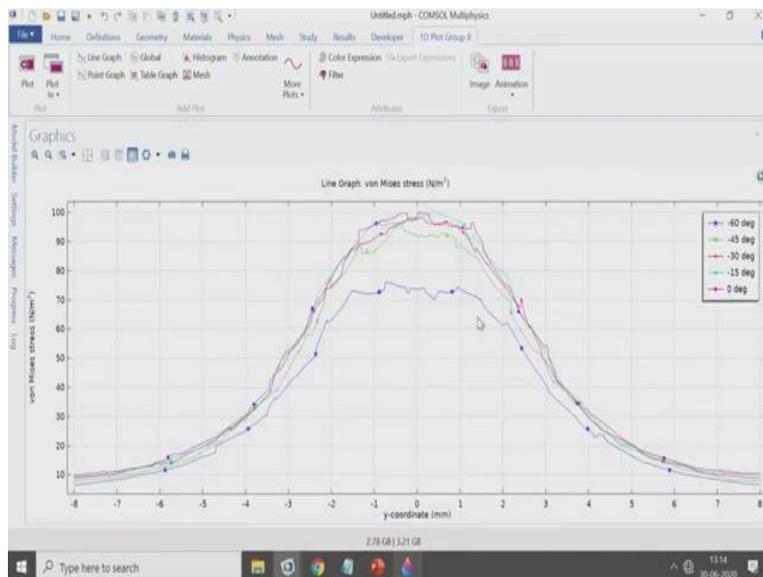
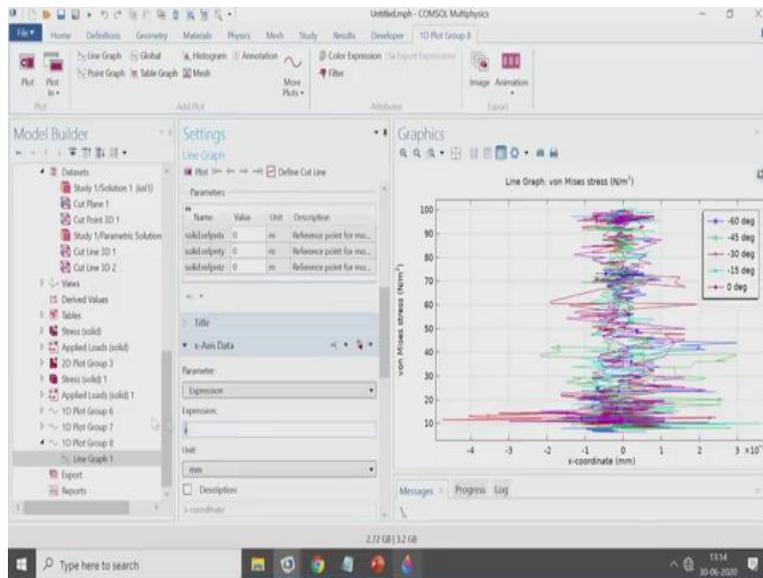
(Refer Slide Time: 38:06)











But this may not be the complete story, to know the complete story you just do not need to evaluate just below the tissue, but also on the sides. For example, you want to now evaluate the stresses within the tissue on a line segment. So how do I do how do I do it? I just go to data set, I right click on the data set, and I use something known as cut line 3D. In the cutline 3D, I use the point as in X axis I use from minus 5 to plus 5.

So it will be from this point to this point and Y axis and Z axis I just put it 00. So now my cut line is somewhere over here. I need to bring a little bit down. So I use minus 2 in both the Z axes. This looks good. This is what I wanted. But I want to have a little bit more in X axis maybe minus 8 plus it would go do it. Yeah, it looks good. So now you can see that your line segment

which is in red colour is where it is positioned within the tissue and within this tissue, I want to now evaluate the stress.

What is the first figure told us is that as you go on in increasing the angle of indentation the stress is being reduced but now let us see, so this is stress below this tissue how it is reducing. Let us see if in the sides, if the stress tissue in within the tissue, if the stress are increasing or not. To do that I want to plot one more 1D plot group.

So I add the results, right click on the results, 1D plot group and in the 1D plot group right click, I use the line graph. In the line graph, I can use any of the lines that is already available or I can use in the data set, cut line 3D 1. So, let me just see in the cut line 3D 1 Yeah, I need to change in this data set as study 1 parametric solution. So now this is the graph, how it looks like.

Rather than solids, dot disp that is talking about displacement, I can use the mises that is the stress being developed for each of the angle of indentation. This graph may look a little bit complex but let us go and the Legends. So now it looks little bit better. You can also add the coloring and style, marker may be as cycle so that all of them have different angle of, different markers.

So from those graph, what do we get to know? So little bit to decrease the complexity what I will do is instead of all I can use from the list and I can choose from 0 to 60 degrees and I click on plot. So what is happening over here? So as we go on we see that the for 0 degrees, 0 degrees you can see again the stress is very high at the center.

So one more addition, what I can do is instead of the X axis right now it is starting from 0 to 16 I can write it as expression and expression as x and click on plot. With this I will have 0 in the center and then offset of plus minus 8 on the X and Y axis. So you can see that I have at the center over here at x is equal to 0 that is if you see the cut line over here that is at this very particular point. At this particular point and minus 8 is somewhere over here.

Plus 8 is somewhere over here. So what do I see over here? That if you remember earlier the point below the tissue, if you increase the indentation that is actually decreasing. Over here you can see that in line segment at 0 degree the highest is somewhere over here. But at minus 15

degrees it shifted towards the left and similarly for, for the plus 15 degrees, so that is when the force is coming like this.

So it shifts towards the left. Then similarly for 30 degrees, 45 degrees, 60 degrees, so it is actually shifting towards the left. So when the force is applied from the right side, the stress is now building towards the left side and for 60 degrees you can see that there is also second wave of pressure that is being developed and this is a symmetrical profile.

So if you choose to plot from minus 60 to 0 degrees you will see that the force is now so what you are right now plotting is a symmetrical graph, so earlier it was on the left side now it shifts towards the right side. So again for 0 degree it is highest in the center but at 1 degree, sorry at minus 15 degrees that is the cyan color it is very high. Minus 30 degree it is even higher. At minus 45 degrees it is even higher at around 1mm and then at minus 60 degrees it reduces but then there is a one more peak on the left side.

So with this quantification you can try to understand which angle of indentation is going to be better. In this case it looks like at 0 degree that is normal indentation, the stress is going to be least with the help of this one also and with the help of these figures. So you can see that, in this case the distribution of stress is actually more flat. It covers more uniform area.

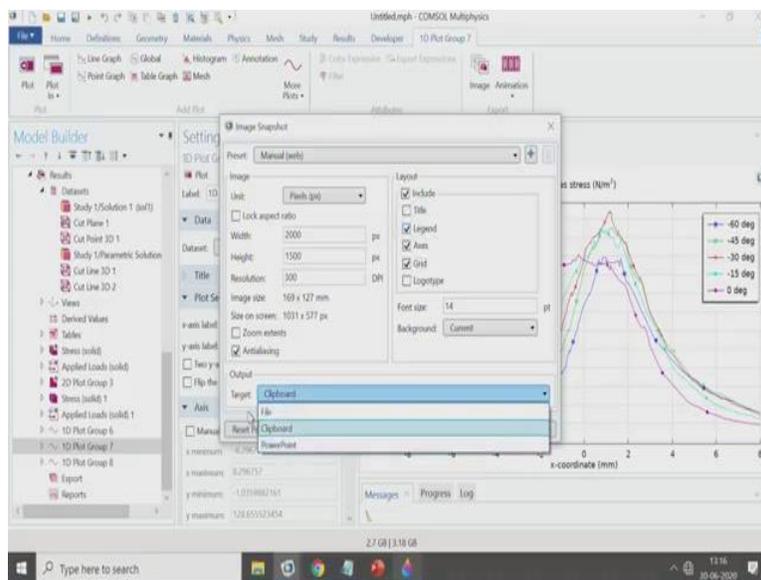
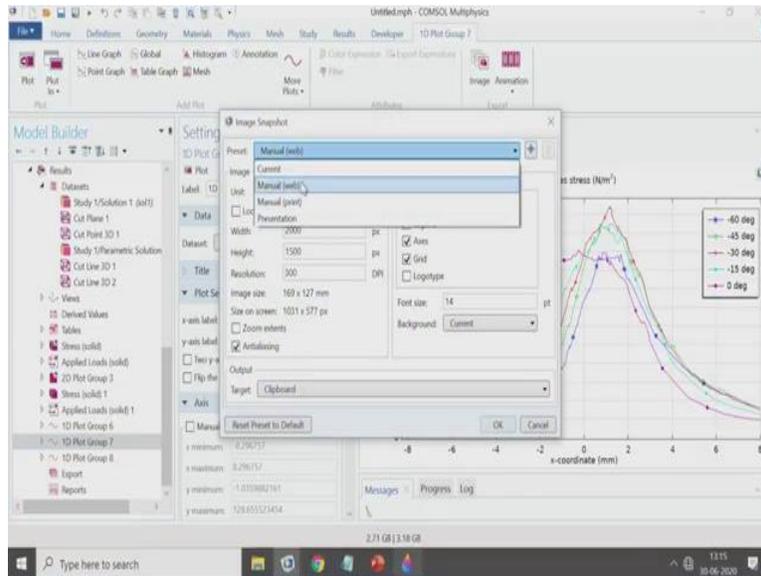
Maybe the integration of this area would be the same as compared to the other degrees but it has a more flatter region. That means that the stress is going to be reduced and the area of contact is also higher. So the pain would be lower. But in the case of angle of incidence to be more, like minus 15 degree, 30 degree, 45, 60 degrees, as you can see that the stress actually increases as compared to the 0 degree.

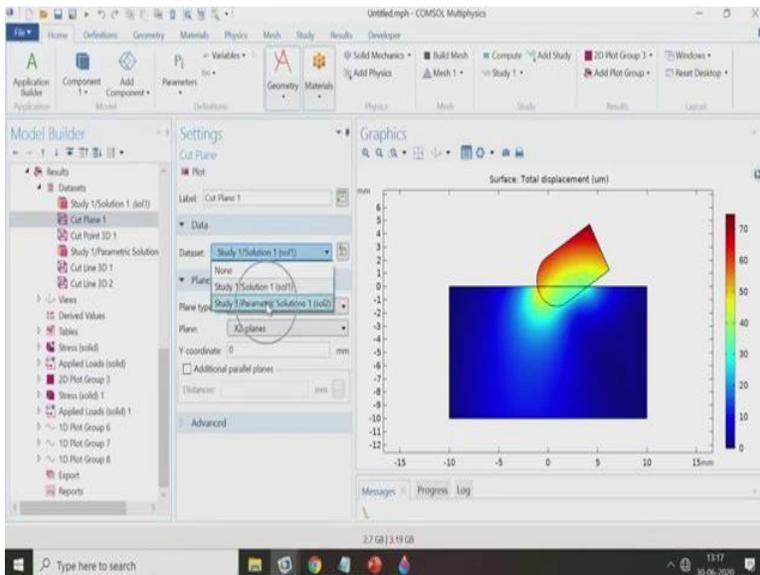
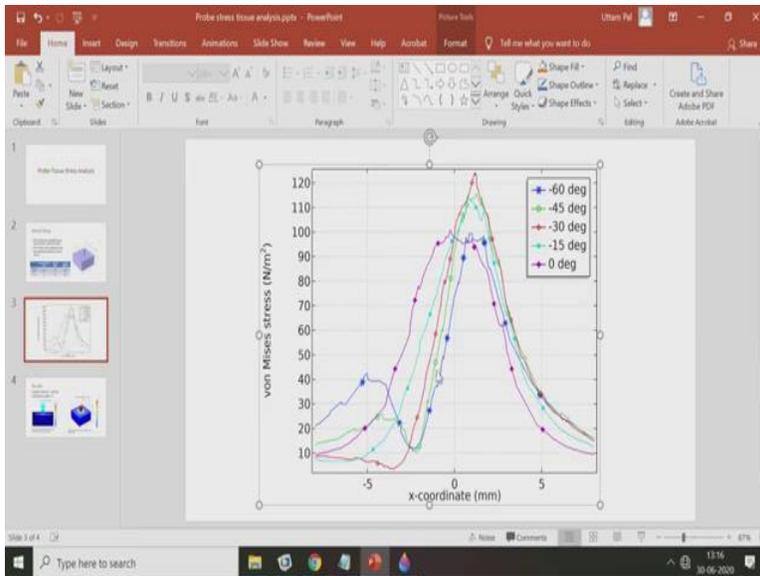
So it seems to be that it would be that it would these angles will be more painful as compared to your 0 degree that is normal angle of incidence. So this kind of analysis you can actually perform. In addition to it, what we evaluated was this axis. You can also evaluate in this axis. So how do I do that? I again use one more cut line 3D 1 or I can just duplicate this cut line and instead of minus X to plus X I can write it as 0 comma 0. I will write Y from minus 8 to plus 8.

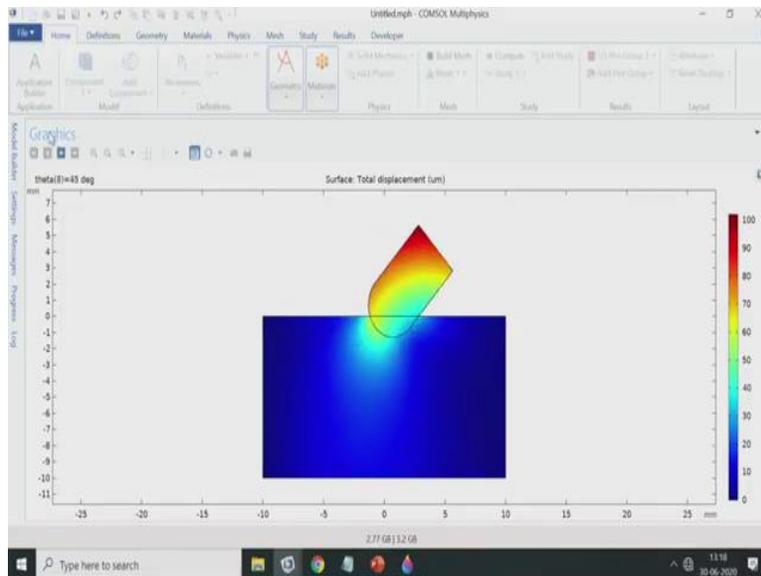
So now you can see that the line has now shifted towards the Y axis. So I can just duplicate this 1D plot group 7 for example. And instead of cut line 3D 1 I can use cut line 3D 2 and I can click

on plot and over here I need to use the expression as y. so now this, from this what do we get to know? We get to know that from around, along the Y axis, the stress is highest of almost all this angles. Only at 60 degrees the along the Y axis it is actually the lowest. So this is what we get to know.

(Refer Slide Time: 47:12)







Now for example you want export any of these figures in your journal papers, for example you want to publish it in your papers. So how do I do that? One of the way is to use the image snapshot and this the best way to do and go ahead with. So just click on image snapshot and there are different features available, so best one is what I prefer is to use manual web and use the width as 2000, height as 1500 and the resolution.

So this is very important when you try to publish, you use the resolution as 300 DPI. This is minimum requirement for any figures to be published. However when you try to increase your resolution to 300 you need to also increase the font size, otherwise the fonts becomes very very small and over here which all, which what are the components of the layouts that you want to select, you can just select.

As of now I just want to have Axes and the Grid points along with the Legends for example and I can just copy it to the clipboard or the file or PowerPoint. As of now I will just copy it to the clipboard and then I can just go to my PowerPoint and for example and just paste it over here. If I want I can even increase my width ratio here like 2500, so it will be more long as compared to the height. So now you can see that it is more long alright.

You can also evaluate the for example you have the stress profile over here. I can first have it made it as parametric sweep so that I have for all the angles. Now what I want to do is, I want to create an animation out of it. It is very simple to create animation. You just need to go to this 2D

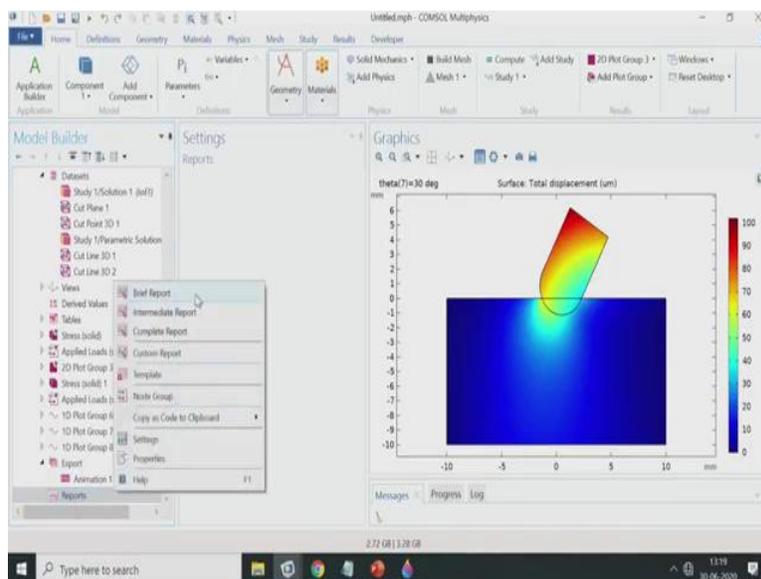
plot group 3 for example if I want to make the animation of this one, I can use the animation. Just click on animation and use the player.

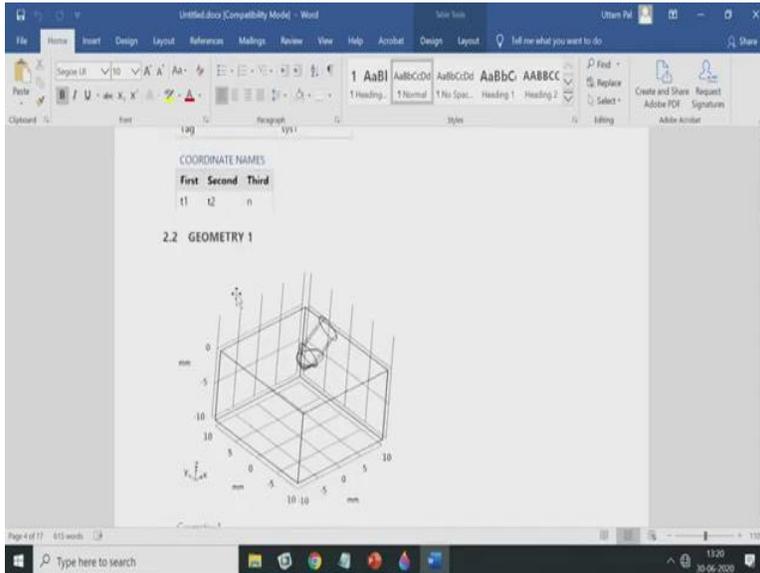
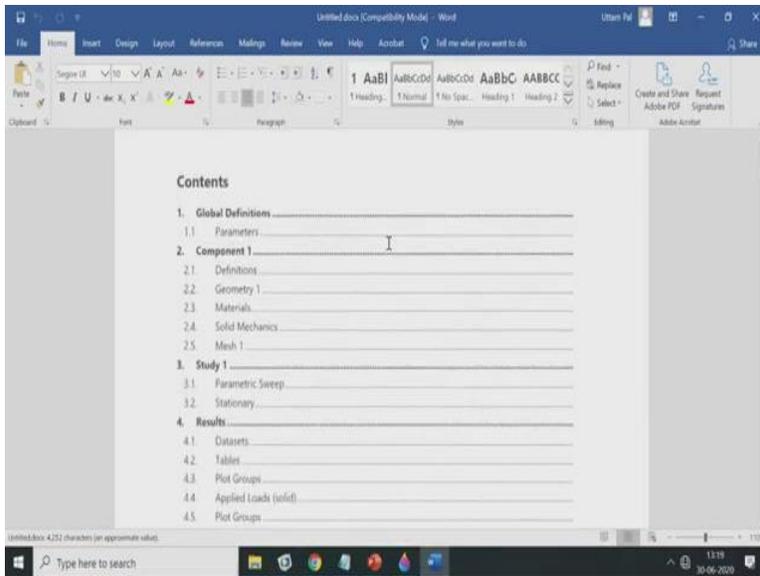
So and then if you want to convert it into a file and then use it into your PowerPoint, just change it to file and this will create a GIF file of the animation and that you can actually import it into a PowerPoint. If you use a player you can choose the player option over here, how much time do you want to devote for each frame. There are 9 total value of sweep so we use 9 number and then we have the play button.

So you can see that for each option you are able to see. If you also click on repeat so that this will actually get repeated again and again. From this you can also see that how the stress or the displacement are moving from left to right or right to left as the probe is moved from left to right. So it is a opposite or a kind of anti-symmetric behavior of the stress profile.

So when the probe is pointing towards the right side, the stress is more over here and when the tissue is, when the probe is pointing towards the left side, the stress is more towards the left. So such kind of analysis, you can actually perform and quantify your results.

(Refer Slide Time: 51:18)





Fixed Constraint 1
Boundary Load 1

2.4.1 Linear Elastic Material 1

EQUATIONS

$$0 = \nabla \cdot \mathbf{S} + \mathbf{F}_v$$

$$\mathbf{S} = \mathbf{S}_{el} + \mathbf{C} : \boldsymbol{\varepsilon}_{pl}, \quad \boldsymbol{\varepsilon}_{pl} = \boldsymbol{\varepsilon} - \boldsymbol{\beta}_{pl}$$

$$\boldsymbol{\varepsilon}_{pl} = \boldsymbol{\varepsilon}_0 + \boldsymbol{\varepsilon}_{el} + \boldsymbol{\varepsilon}_m + \boldsymbol{\varepsilon}_{sh} + \boldsymbol{\varepsilon}_p + \boldsymbol{\varepsilon}_c + \boldsymbol{\varepsilon}_{cp}$$

$$\mathbf{S}_{el} = \mathbf{S}_0 + \mathbf{S}_{el} + \mathbf{S}_1$$

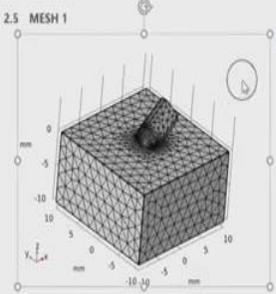
$$\boldsymbol{\varepsilon} = \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T)$$

$$\mathbf{C} = \mathbf{C}(\mathbf{E}, \nu)$$

Page 6 of 17 615 words

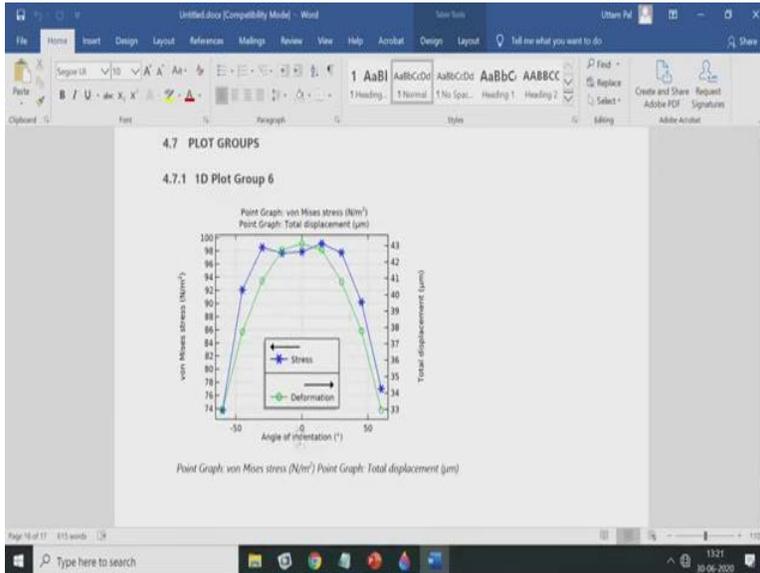
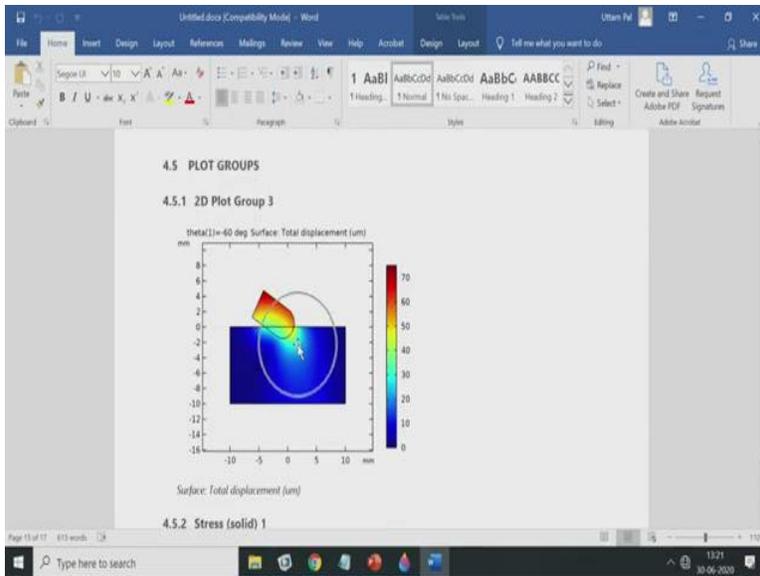
$\mathbf{S} \cdot \mathbf{n} = \mathbf{F}_s$
 $\mathbf{F}_s = -\rho \mathbf{g}$

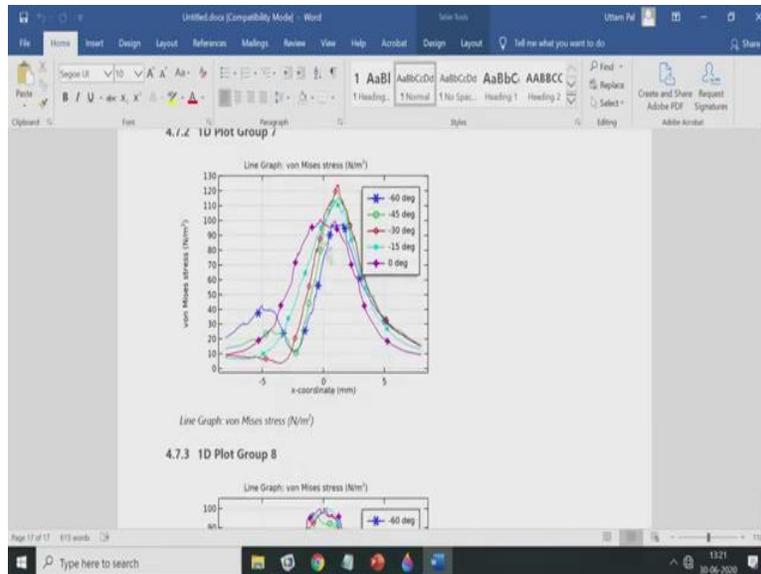
2.5 MESH 1



Mesh 1

Page 7 of 17 615 words





You can also create reports. To create report, just right click on the report, brief report for example. I just wanted to create a brief report. So just create a brief report and just write, just click on Write. So it will create a word file of your report. If you want you can also have a HTML file. So now you can see this, the report in word file has been created and it has all the parameters that is used to model.

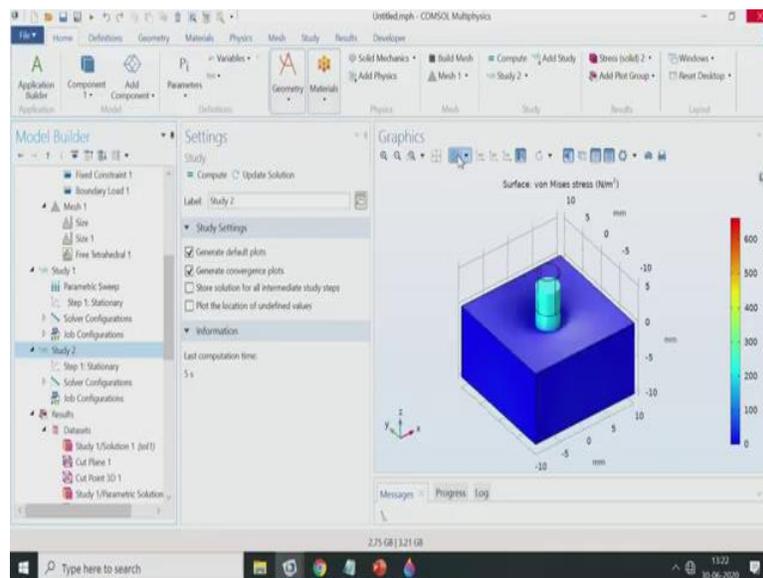
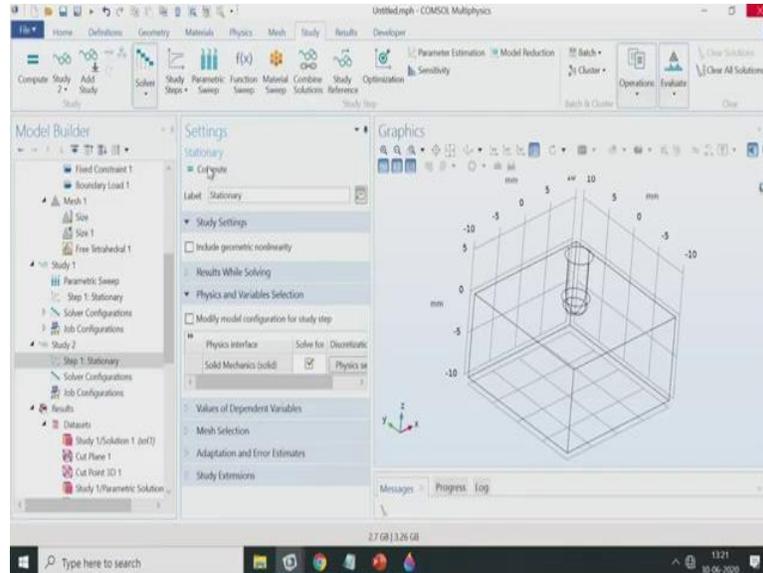
It has all the definitions, the geometry that you have made for all the entity, what are the material properties, which physics that you, have selected, what equations the physics have solved. So do not need to write those equations from scratch and just directly use this. Just maximize this and try to use it.

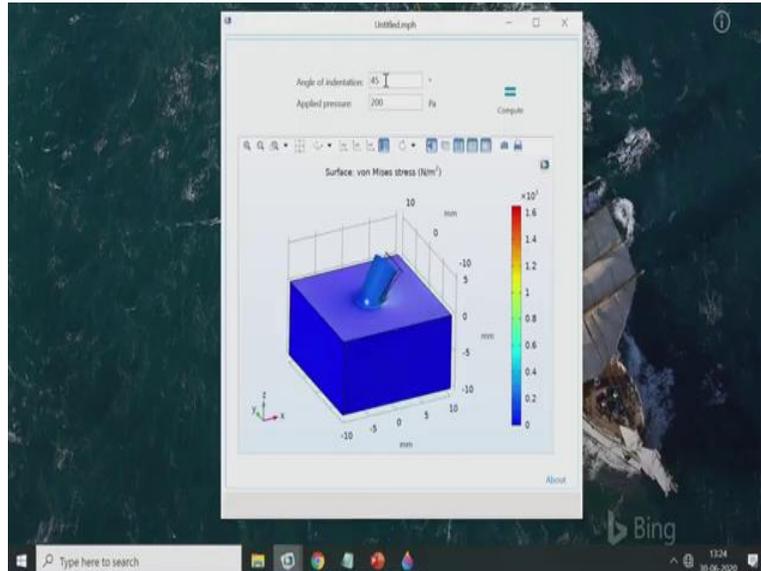
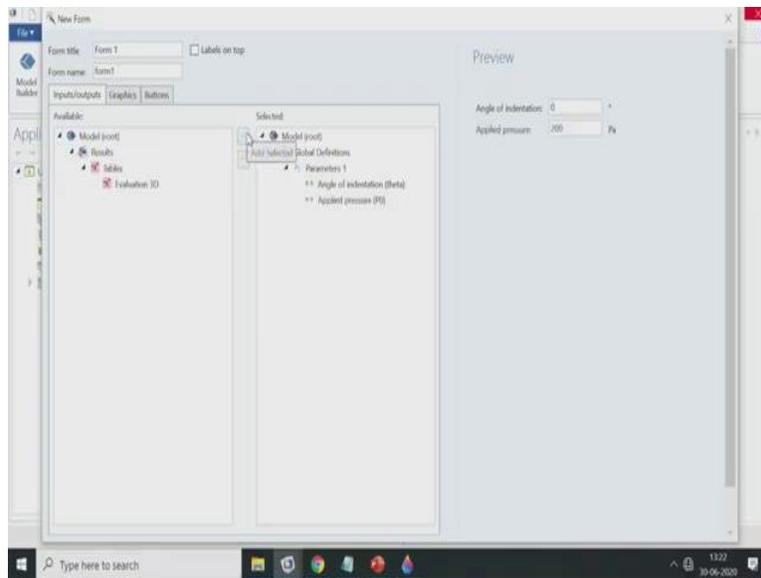
What are the boundary conditions that you applied that is fixed constants and boundary load. The meshing that you have used and then which kind of study have you performed, what parametric sweep have you performed and then finally the results. This one talks about the different-different data sets that you have created and then finally you have the results.

So this the von Mises stress on the complete volume, how the load is applied, you can see that load is applied normally to this electrode and then you have the results in the 2D plot group, 3D plot group, again the force and then you have the results in the 1D plot group, how the stress and the deformation changes below the tissue, below the probe within the tissue as the function of angle of incidence, indentation. And then for different-different angles how the stress profile

looks like and in both X and Y axis. So you can use this report, share it with your colleges or your professors if you are a student and if you are a professor you can share it with your students.

(Refer Slide Time: 53:42)





One more way to portraint this App if you are using the form of a App, for that I will just create one more study, a very simple study and I click on compute. Because I have performed a parametric sweep in my study so to simplify I will just add it a one more study.

Now if I want to make a, if I want to share this particular App with someone who does not know COMSOL, in that case it is very simple to create Apps and this App will give you all the parameters that you want, just change the parameters and you can see the results very quickly. So how do I make an App? So just click on App builder over here on the left side bottom and this option over here to make a new form.

So just click on new form and here are the parameters which I want them, my colleague or my student or my professor want, I want them to change, so add those parameters. Then I go to the graphics, what graphs do I want them to see? For example I want to see, I want them to see this stress profile and then the buttons. I want them to compute my study 2 which is a simplified study without the parametric sweep so just click on study 2.

On the right side what you see is the, a preview of your graph or your form. So just click on okay, just make it a little bit larger and make the compute on the right side. Somewhere over here understood and then I just click on test App. So this is how the App looks like. So this App is, now you can change the angle of indentation for example I can choose 45 degrees and I click on compute.

So now it will get computed, it will take the same time that it is usually going to take but now you have the results for the 45 degree. So it is App is now very simple. You can share it with your colleagues, you can also plot, use the 1D plot group, the point graph plot and other results and you can share it with your colleagues. They can open in your mobile phones. They can compute this model in the mobile phones for example.

You can also use the compiler license of COMSOL to make this App as a dot exe file and share it to many people that you want, share it across the world and if you are doing a consultancy business then you can also sell this App. This is how this use of app can help you to proceed with your models and sharing of your models. So that is it with the discussion of the model. If you have any questions please do not hesitate to write to us. We will definitely get back to you. Thank you.