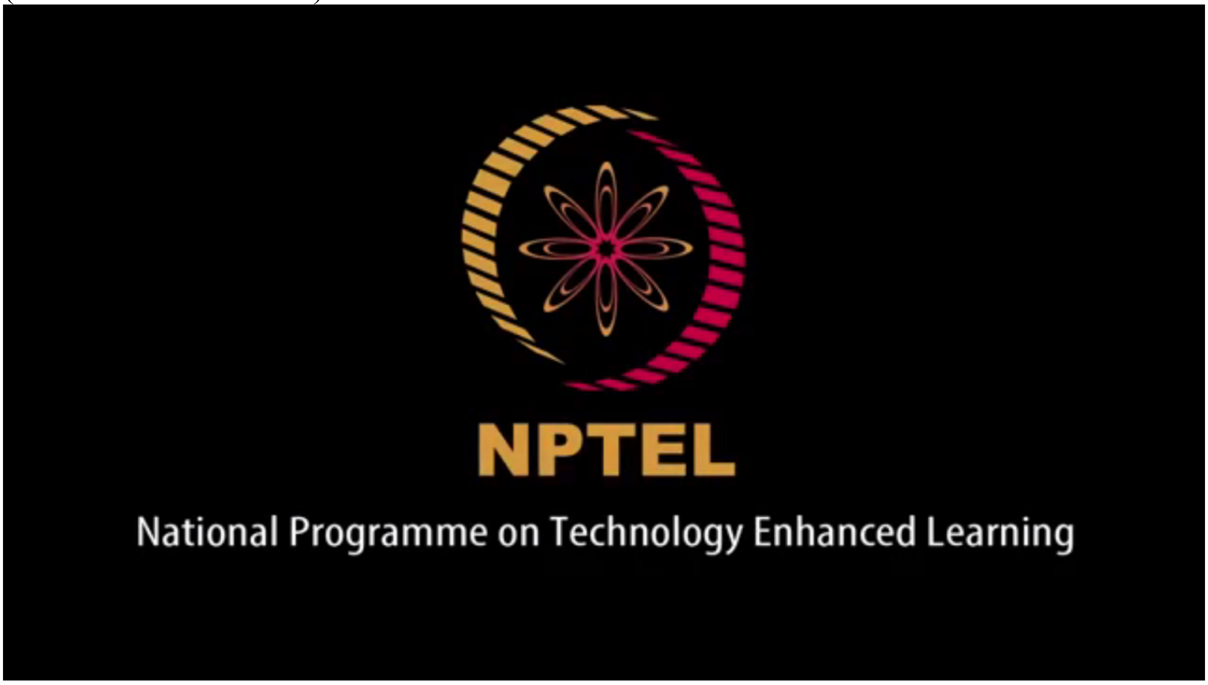
(Refer Slide Time 00:00)

Indian Institute of Science

(Refer Slide Time 00:04)



NPTEL
National Programme on Technology Enhanced Learning

(Refer Slide Time 00:08)

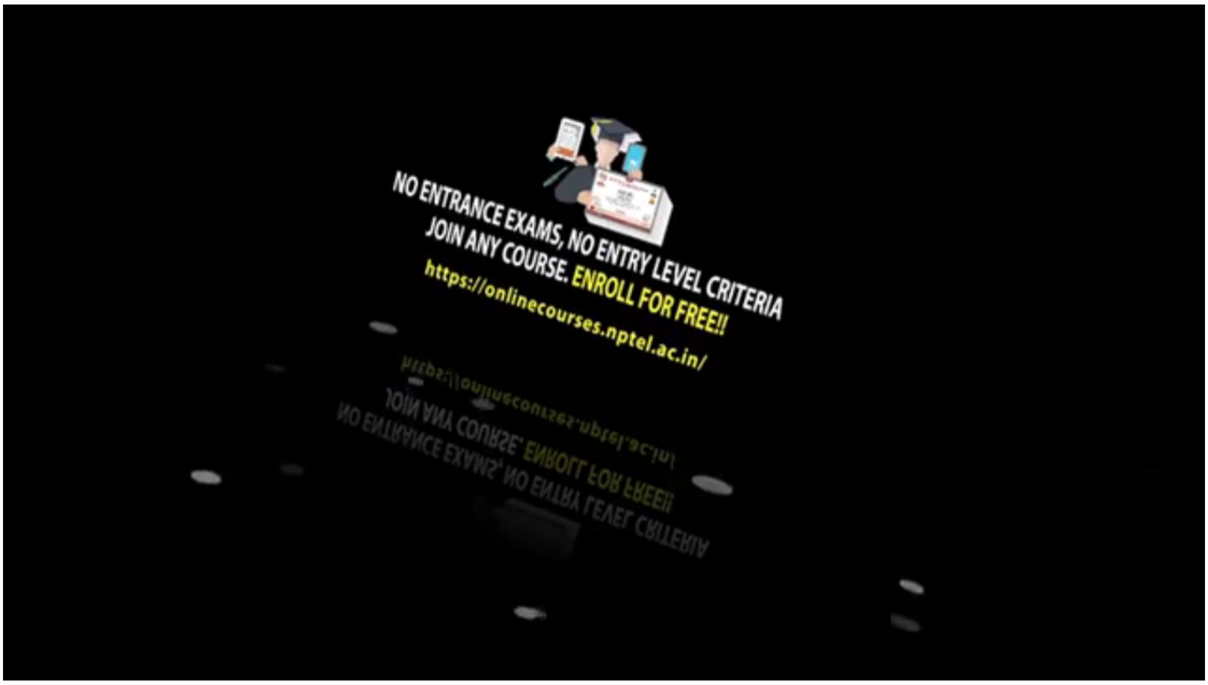
NPTEL IS OFFERING ONLINE
CERTIFICATION COURSES

(Refer Slide Time 00:11)

CERTIFICATES FROM THE
IITs & IISc
are just a click away..

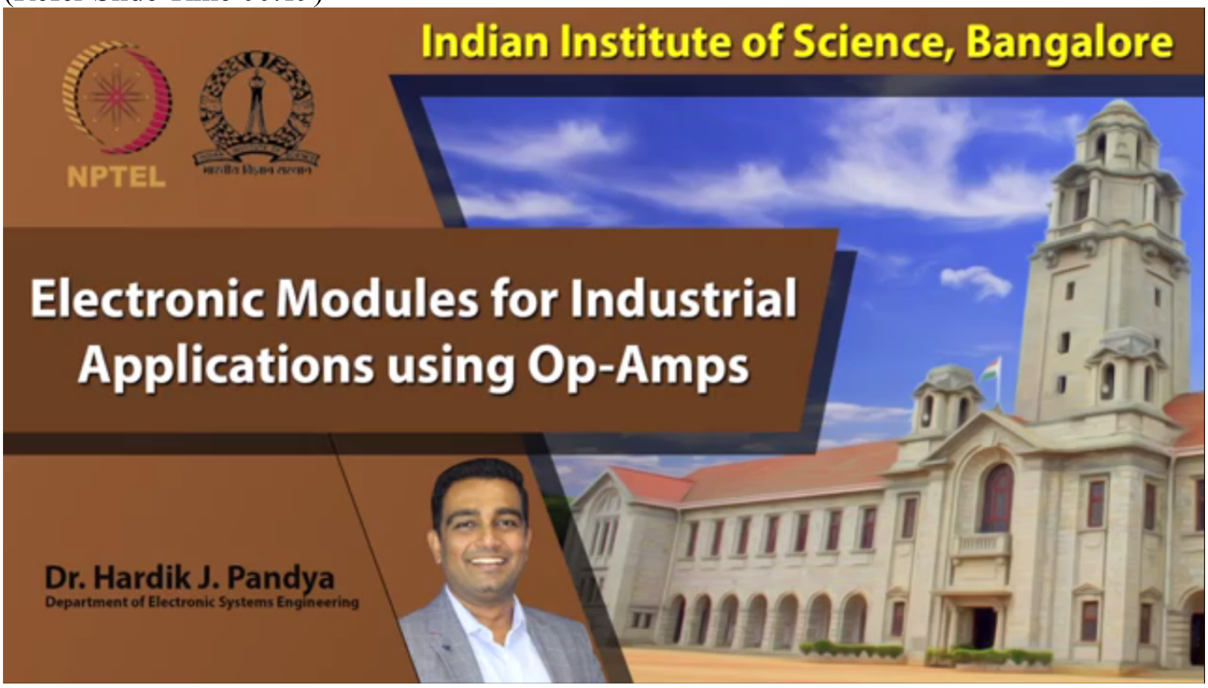
onlinecourses.nptel.ac.in

(Refer Slide Time 00:15)



NO ENTRANCE EXAMS, NO ENTRY LEVEL CRITERIA
JOIN ANY COURSE. ENROL FOR FREE!!
https://onlinecourses.nptel.ac.in

(Refer Slide Time 00:19)



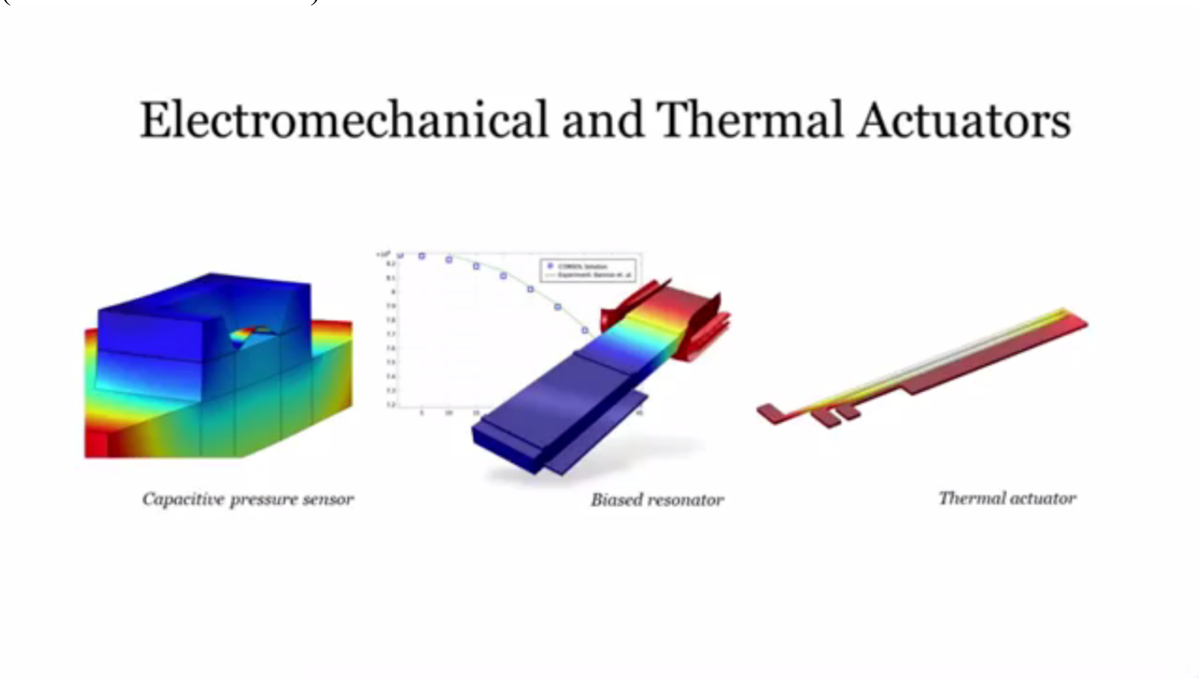
Indian Institute of Science, Bangalore
NPTEL

Electronic Modules for Industrial
Applications using Op-Amps

Dr. Hardik J. Pandya
Department Of Electronic Systems Engineering

So let me continue, let us continue with the electromechanical and thermal actuator.

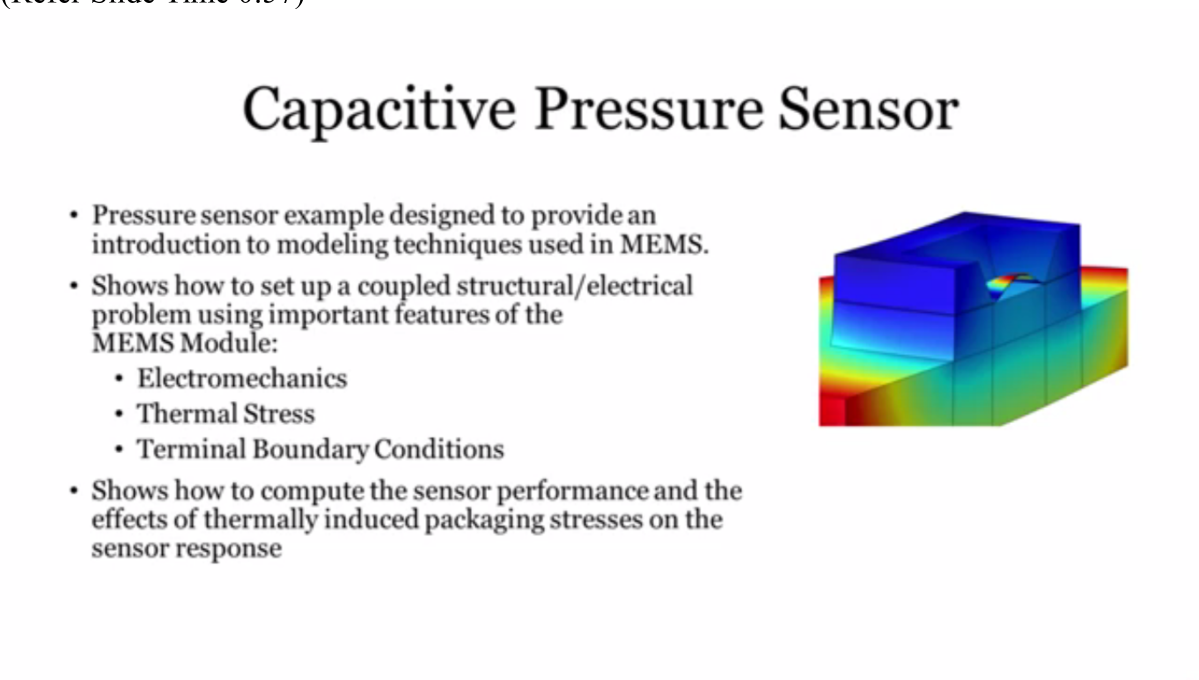
(Refer Slide Time 00:34)



So till now what we saw was piezoelectric materials, how to model that in COMSOL. We also talked about how to model piezoresistive structures within the COMSOL Multiphysics, how the coupling of structure mechanics with the electrostatics was occurring.

In this example that we are going to talk about, we are going to talk about how the actuation actually happens and what are the physics behind that.

(Refer Slide Time 0:57)



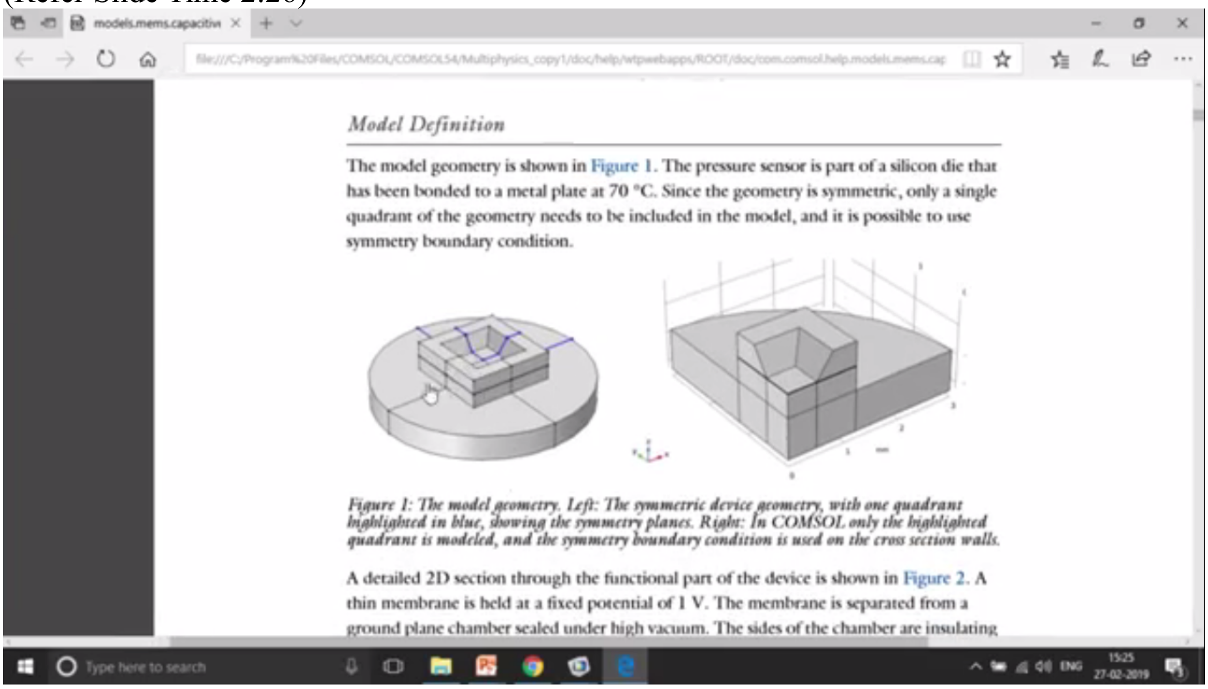
One of the example model that comes into picture is Capacitive Pressure Sensor. These are regularly used in our mobile phones as you know and this is kind of touch sensors that people want to model or they want to have it on their mobile phones.

So let me just go ahead and open this particular model. So I go to COMSOL and I go to File, Application Library, and when I go and search for piezo, sorry, not piezo, this is now electromechanical system. So we have done away with the piezoelectric device. Now we are going to move ahead with the electro-mechanical actuations.

So let me go to actuators and I think it should be somewhere over here or it could be a part of structural mechanics module. So if you are not able to search, just search for it, capacitive pressure sensor. Since we are searching for the first time, it may take some time, but then it will take very quickly.

So this is the example which was already available in MEMS module sensors. So, capacitive pressure sensor, right. So let me just open this documentation to help you understand the Physics part.

(Refer Slide Time 2:26)



Model Definition

The model geometry is shown in Figure 1. The pressure sensor is part of a silicon die that has been bonded to a metal plate at 70 °C. Since the geometry is symmetric, only a single quadrant of the geometry needs to be included in the model, and it is possible to use symmetry boundary condition.

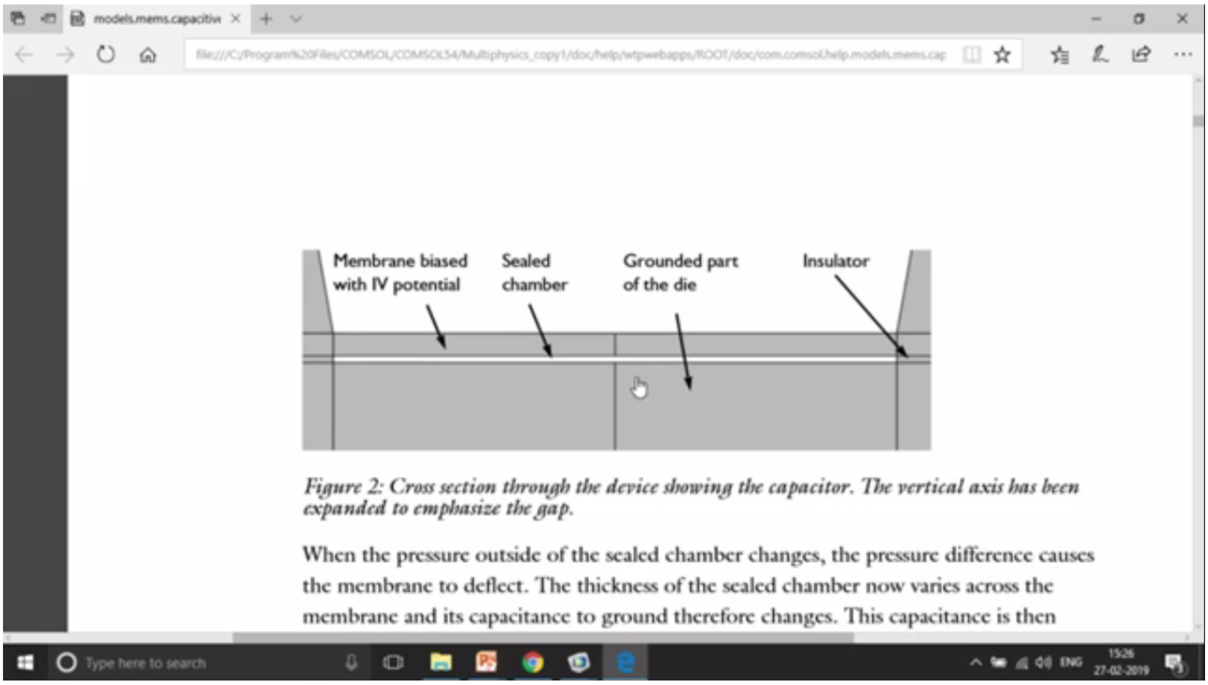
Figure 1: The model geometry. Left: The symmetric device geometry, with one quadrant highlighted in blue, showing the symmetry planes. Right: In COMSOL only the highlighted quadrant is modeled, and the symmetry boundary condition is used on the cross section walls.

A detailed 2D section through the functional part of the device is shown in Figure 2. A thin membrane is held at a fixed potential of 1 V. The membrane is separated from a ground plane chamber sealed under high vacuum. The sides of the chamber are insulating

So, again, that you see over here, this is the actual geometry that you want to model. Okay. So as you can see over here, this is a circular geometry, which is very nicely symmetric in nature and we actually exploit the symmetric nature of your geometry. What we do over here? We model only a quarter part of your geometry instead of the complete spherical or the cylindrical geometry.

So this is the actual domain which is model in COMSOL. So if you take a cross-section over here, you will see something like this, right?

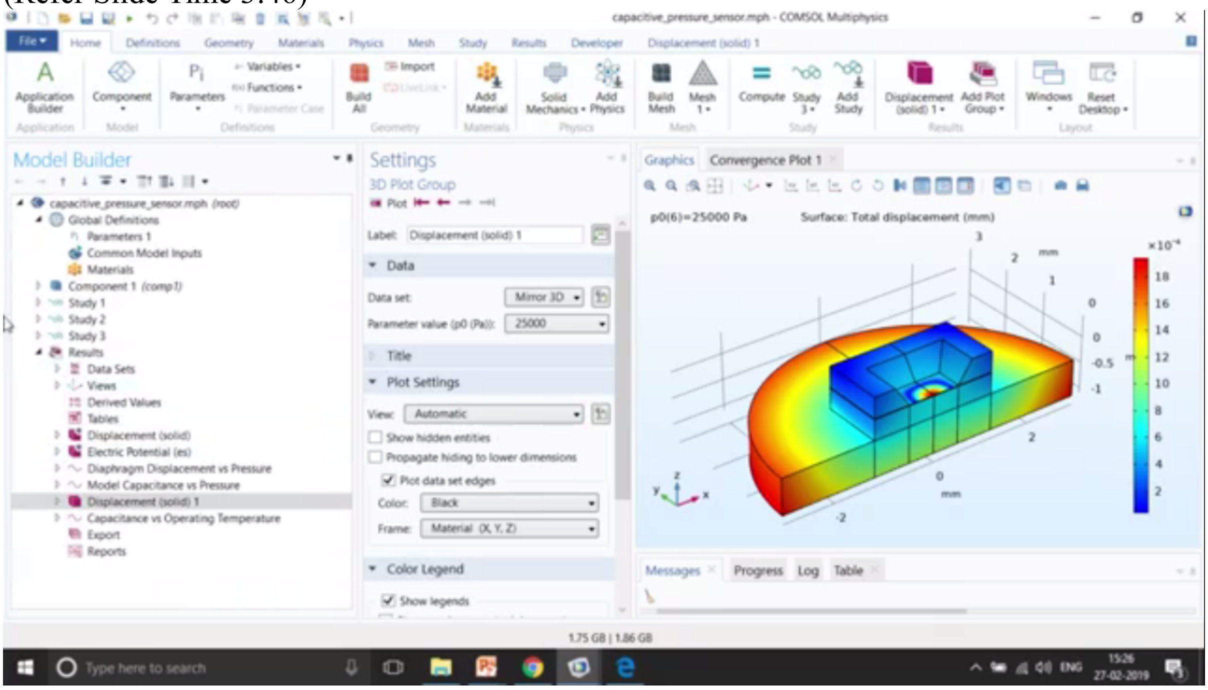
(Refer Slide Time 03:07)



So you have the sealed chamber in the middle, and then you have insulators on both the sides, and then there are two membranes. One is the bottom part and the other is the top part. Okay. And this actually works as a capacitor because based upon the deformation that is going to happen over here, the change in capacitance is going to occur because capacitance is nothing but εA/d. So if the distance is decreasing, your capacitance is going to increase, right?

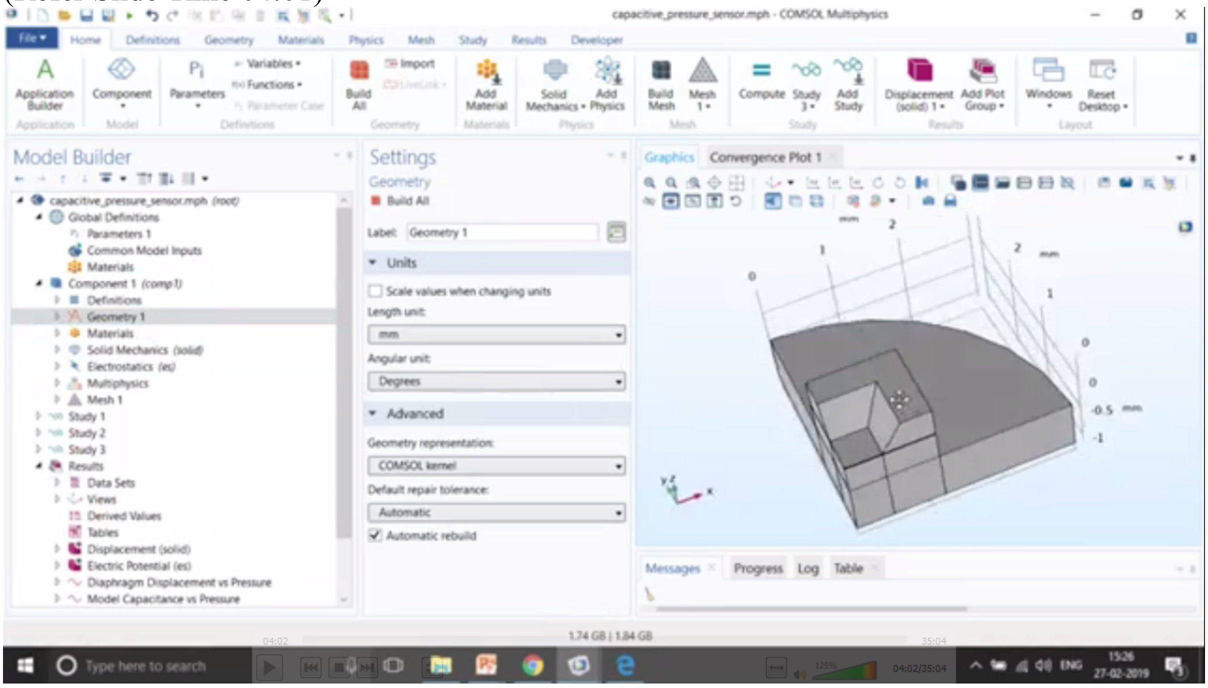
So let us go to the model. So let me open this model.

(Refer Slide Time 3:46)



So over here again, the first part is to make the geometry. So here you can see that how the geometry has been made, first making the blocks, and then partition domains, and then finally we have this particular domain, right?

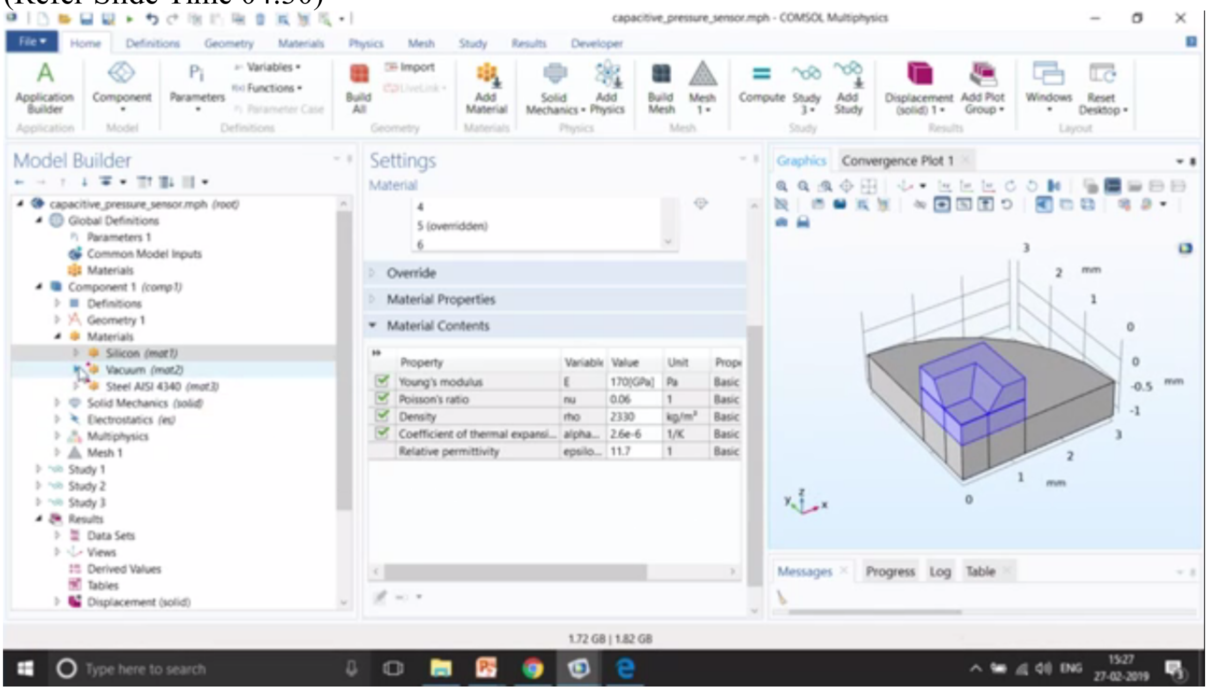
(Refer Slide Time 04:01)



So we are going to model only a quarter part and then we are going to apply symmetric boundary condition on the right and on the left side. Okay.

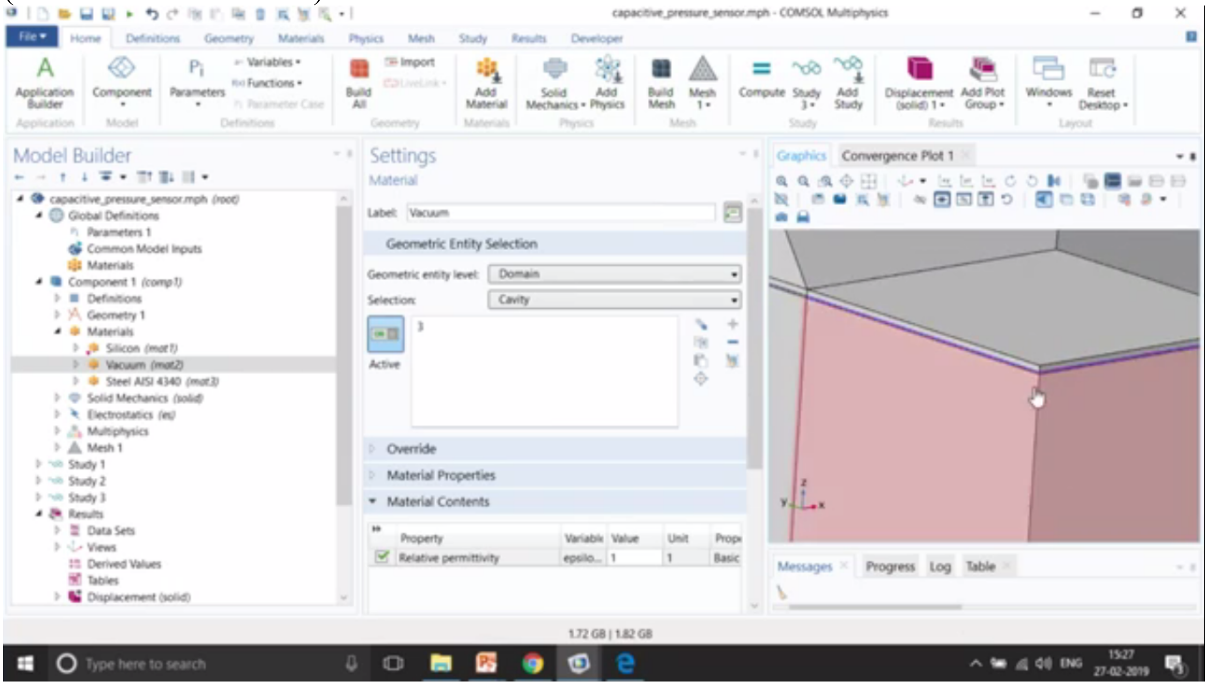
Next part is the materials part, very important part. Here we have used the scalar material properties for the silicon as you can see over here and also the coefficient of thermal expansion comes into the picture. So that means this structure is also dealing with the thermal expansion.

(Refer Slide Time 04:30)



The next part is the vacuum part that happens between the two domains over here. This is the doom that is going to deform to the downward direction and the capacitance is going to change.

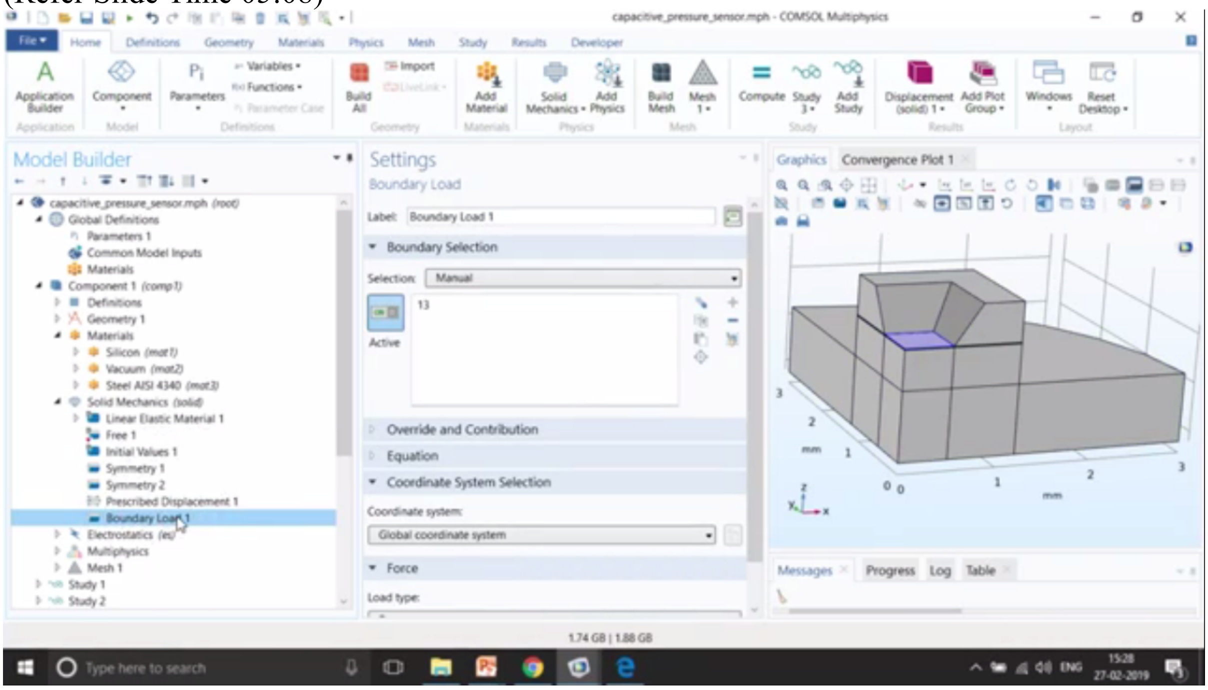
(Refer Slide Time 04:41)



The next part is the steel domain that is in the bottom. This is the major part was steel and this would be again having a very high Young's modulus and very high density. Again, everyone is scalar over here.

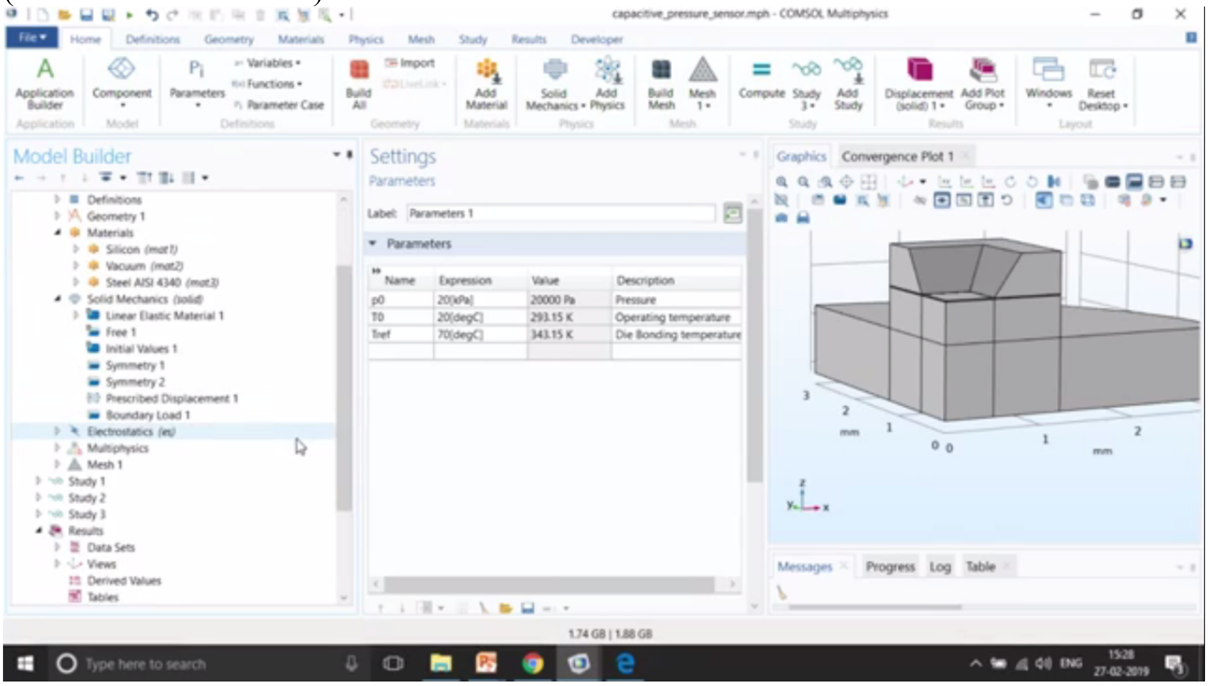
The next part is electrostatic, sorry, the next part is the suction mechanism that would be solid mechanics part, and over here we are giving a boundary load as you can see over here.

(Refer Slide Time 05:08)



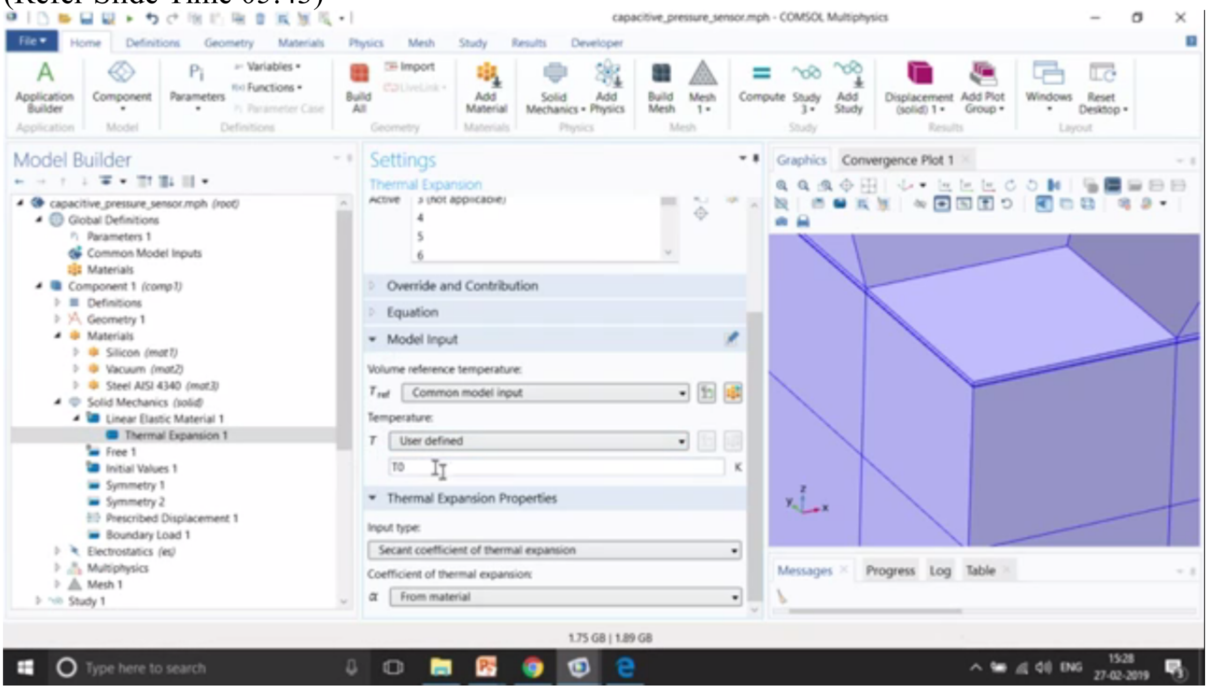
So boundary load has been given on this particular boundary. Okay. In this particular boundary, how much pressure has been given? Here it's p_0 pressure that has been given. A p_0 pressure has been defined over here, 20 kilo Pascal. Okay.

(Refer Slide Time 05:23)



So we give a particular load in the top and it is going to deform because it's an elastic material. So it's going to deform and the along with the deformation, there is also thermal expansion that is because of the temperature, there is going to be some, some external temperature.

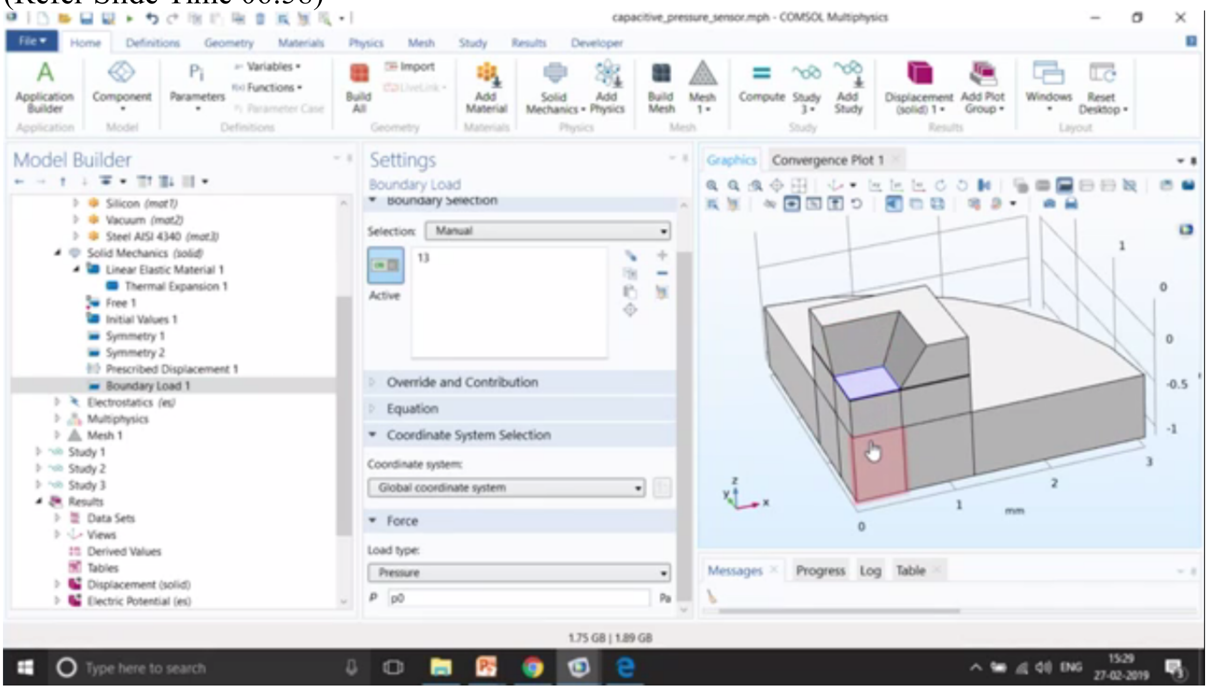
(Refer Slide Time 05:43)



So here the temperature external is T_0. So it's around 20 degrees. This is operating temperature and then we have die bonding temperature that has been given in the model input over here T_{ref}, right? So here this is while the bonding was taking place at that time, some temperature would have been there and because of that there could have been some thermal expansion of your geometry. So it is not going to be, so this is while the fabrication is going to take place, so because of the fabrication issues also some kind of a deformation that takes into picture because of the thermal expansion.

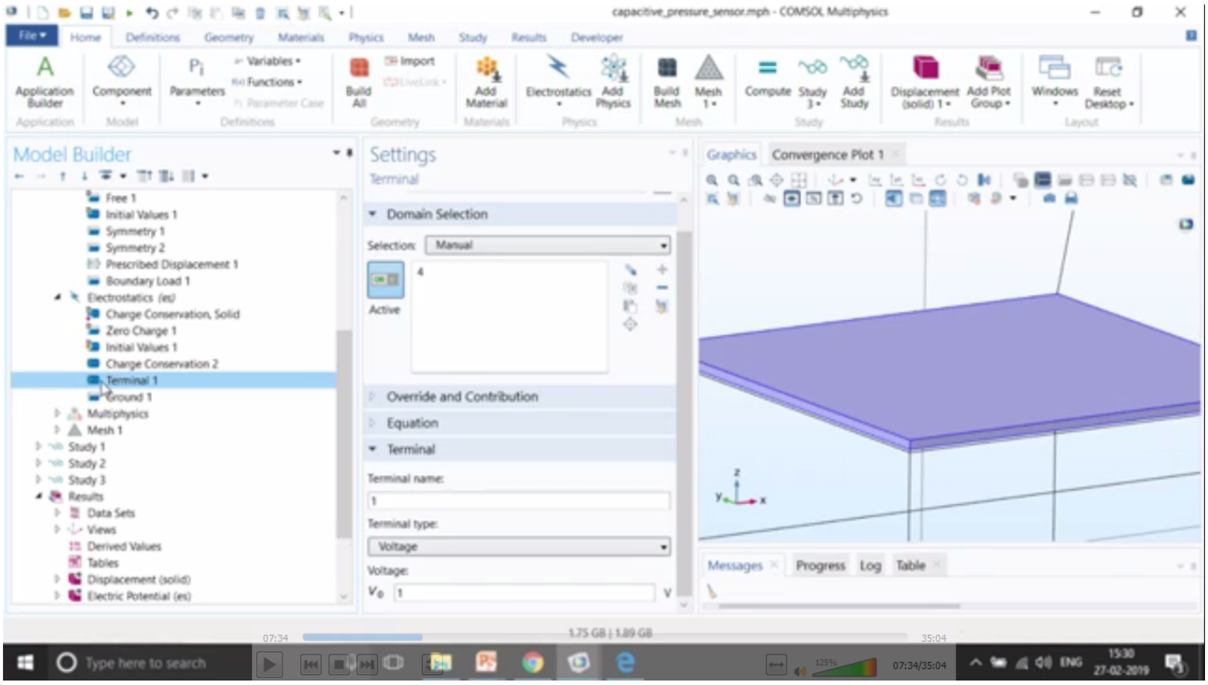
We definitely want to give symmetric boundary condition because we are modelling only quarter part. So we have the symmetry boundary condition on the right side and on the left side, and then we give prescribed displacement. So what, why we are giving it is we are saying that it's not going to move in the top part. So it's not going to move in that direction in this particular point. So it's kind of we are giving some kind of constrains to this particular device and finally the boundary load from the top side. So it's going to give the load from the top to bottom. So it's going to push it to the bottom.

(Refer Slide Time 06:58)



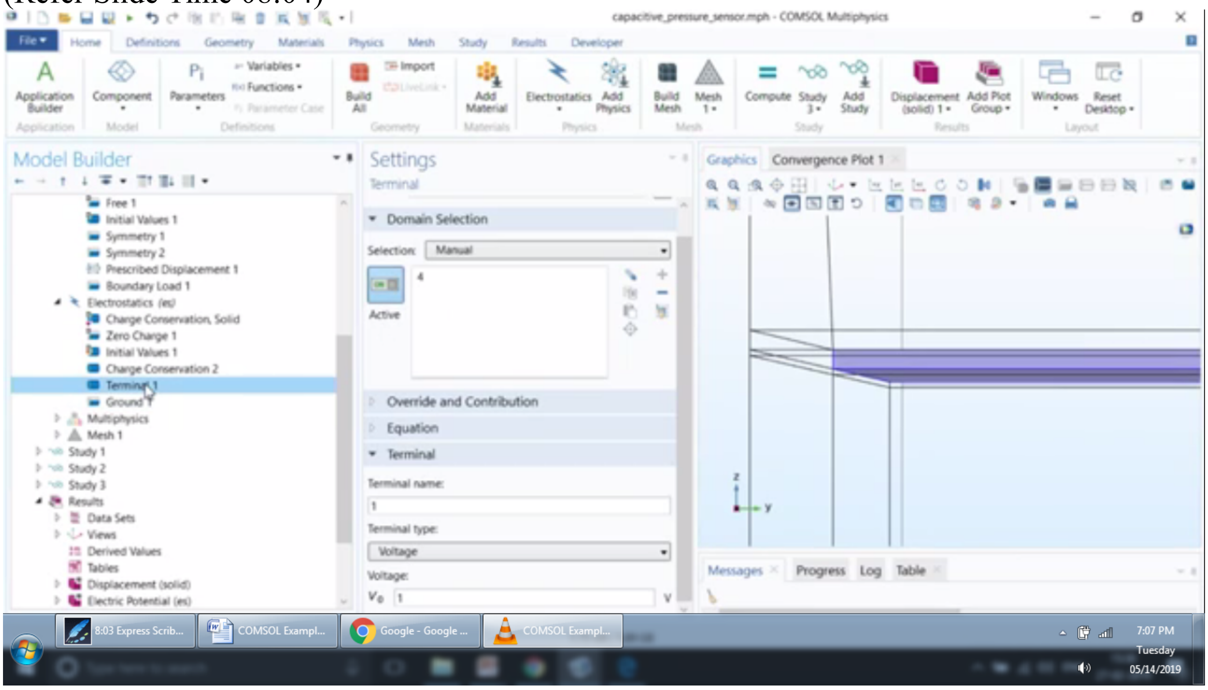
The next part is the electrostatics part and the electrostatics, of course, we are going to give potentials. So in the case that the terminal boundary condition that is giving a potential of 1 volts. That is the top part. Zoom in and you will see that the top part, the whole domain is giving a, given 1 volts and the bottom part that the, so let me just enable the wireframe rendering and here it is a boundary. So you will notice that this is a boundary. This is not domain. However, the terminal is a domain.

(Refer Slide Time 07:33)



So how do you signify what is the difference? Here you will see the blue is completely filled, the cylinder and over here the blue is filled only on the surface of the ground, right? So, but in the middle it is still vacuum, right? So you can see over here, right, so in the bottom you have the ground and terminal is in the top, but in the middle it is still left. So let me just go to have a different views. May be that could help. Yeah. So you can see this is the bottom part and this is the top part terminal.

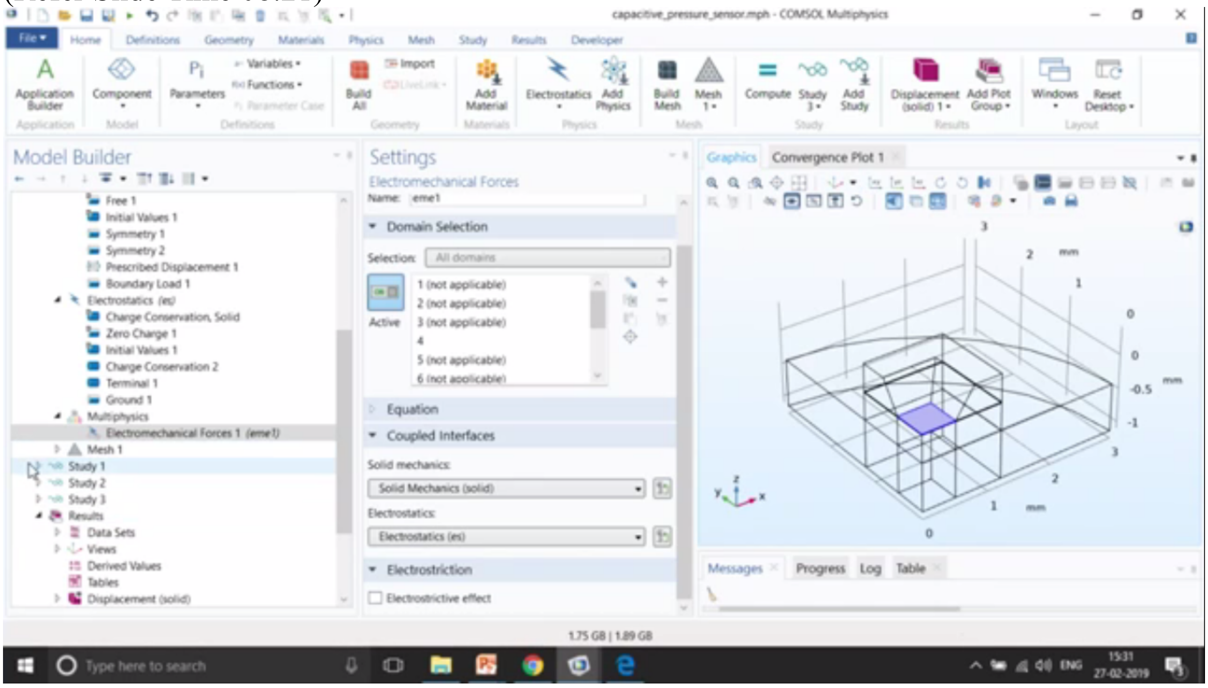
(Refer Slide Time 08:04)



Some part is already there which is missing. Okay. That is the vacuum.

The next is the Multiphysics. So here we move from piezoelectric to electromechanical forces, which covers solid mechanics with the electrostatics. Okay.

(Refer Slide Time 08:21)



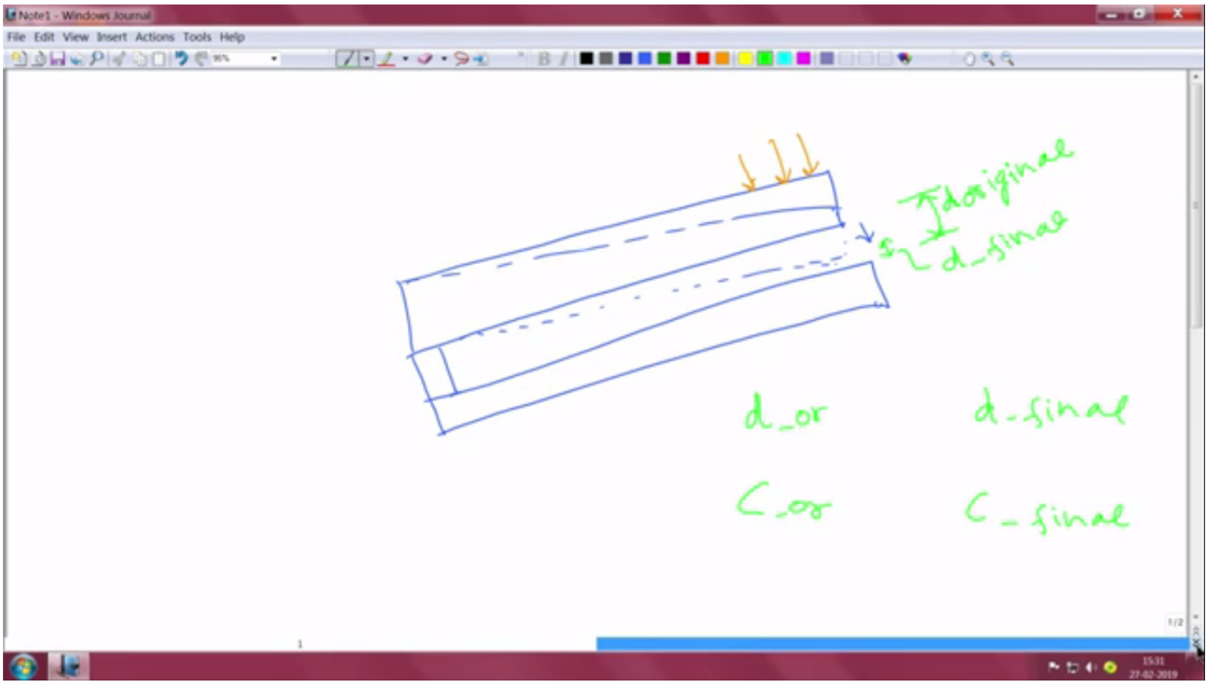
So that’s how it’s done. So it has different types of studies. The first one is solves all, all the three of them as you can see over here: the solid mechanics, electrostatics and the moving mesh. Moving mesh is the most important part.

Now let me give you some introduction of moving mesh. So, for example, you have a particular geometry. So let me just go and show it in the making a figure. So, for example, we have a structure like this. So this is what structure that we have. Okay. And then we have one more bottom structure. Okay. And we are going to actually. We are going to apply the force from the top and which is going to make this structure to bend.

For example, it's going to bend like this, right? So it’s going to bend to the bottom and we have two different values of d, for example. So this was the actual d value that is the distance between the two plates. This is d_original and then we have after the displacement that let us define it as d_final that the position at the final how much is the displacement. So we have d_original and we have d_final, right? So the d_original will give you the original capacitance and the d_final will give you the final capacitance after deformation.

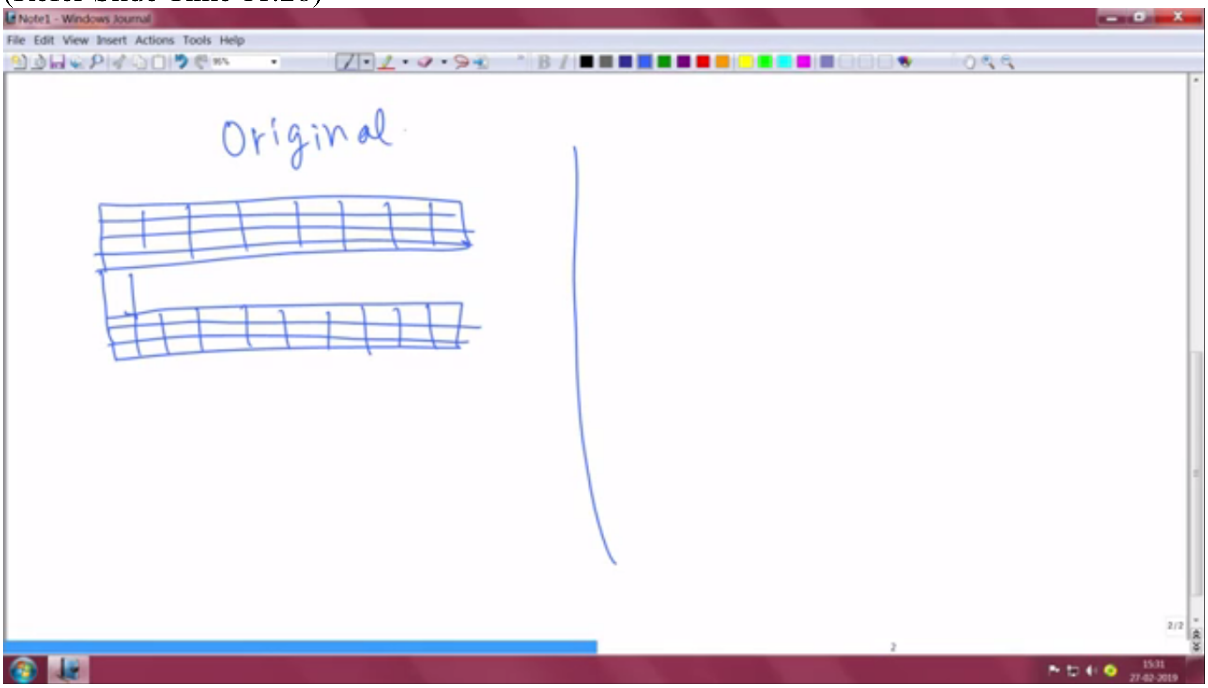
So the most important thing that is happening over here is, is that after the deformation the mesh of your upper cantilever beam also needs to change, right? So if you are meshing in the first part is the straight one, right, so let me go to the next page.

(Refer Slide Time 10:49)



Yeah. So if we talk about two parts, so my first part is going to be the straight part. This is just an explanatory diagram. This is not what is meshed over here. For example, over here we have this kind of meshing. So meshing is like we divide the actual geometry into small, small parts, right? So this is the original geometry. So let me write original.

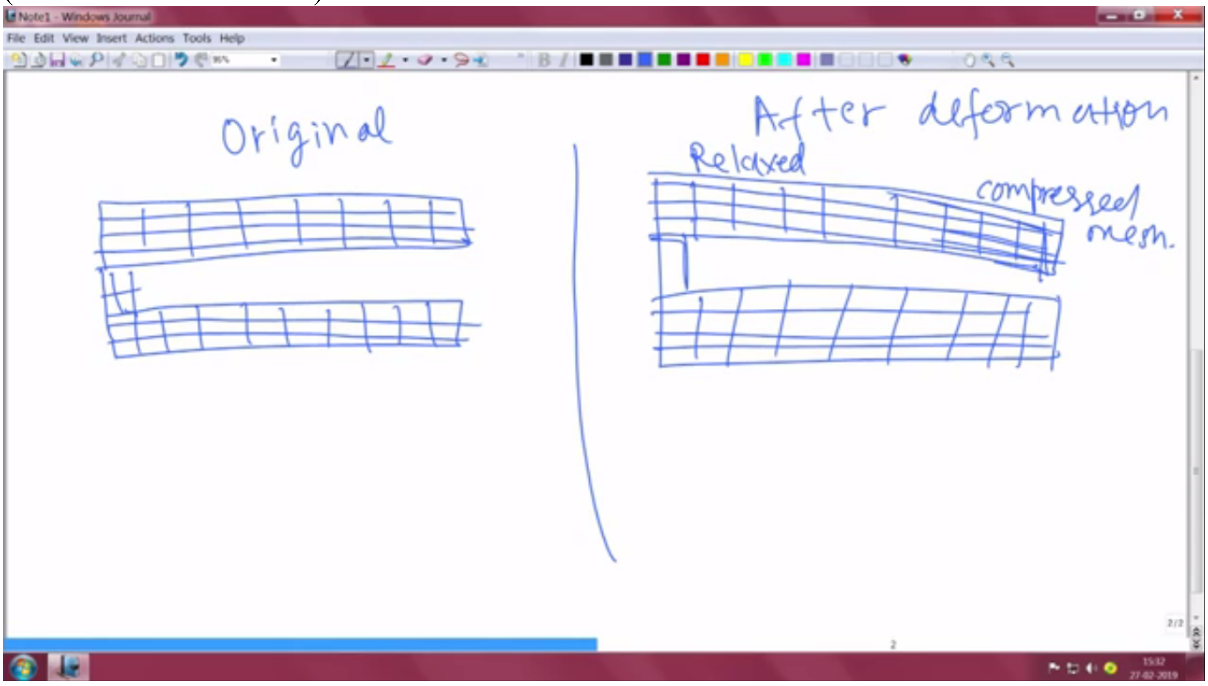
(Refer Slide Time 11:28)



After deformation, once the load is given, so let me write after deformation, the structure is going to actually deform. So by deform I mean the bottom structure is going to remain the same because we have given a kind of a constraint in the bottom, but the top structure is going to move like this. Okay. So you will see that your mesh also needs, needs to get deform, right?

So here you will find that there are more compressed structure over here as compared to the more relaxed versions on the left side. So here it would be more relaxed. Here it would be more relaxed mesh and over here it would be more compressed mesh.

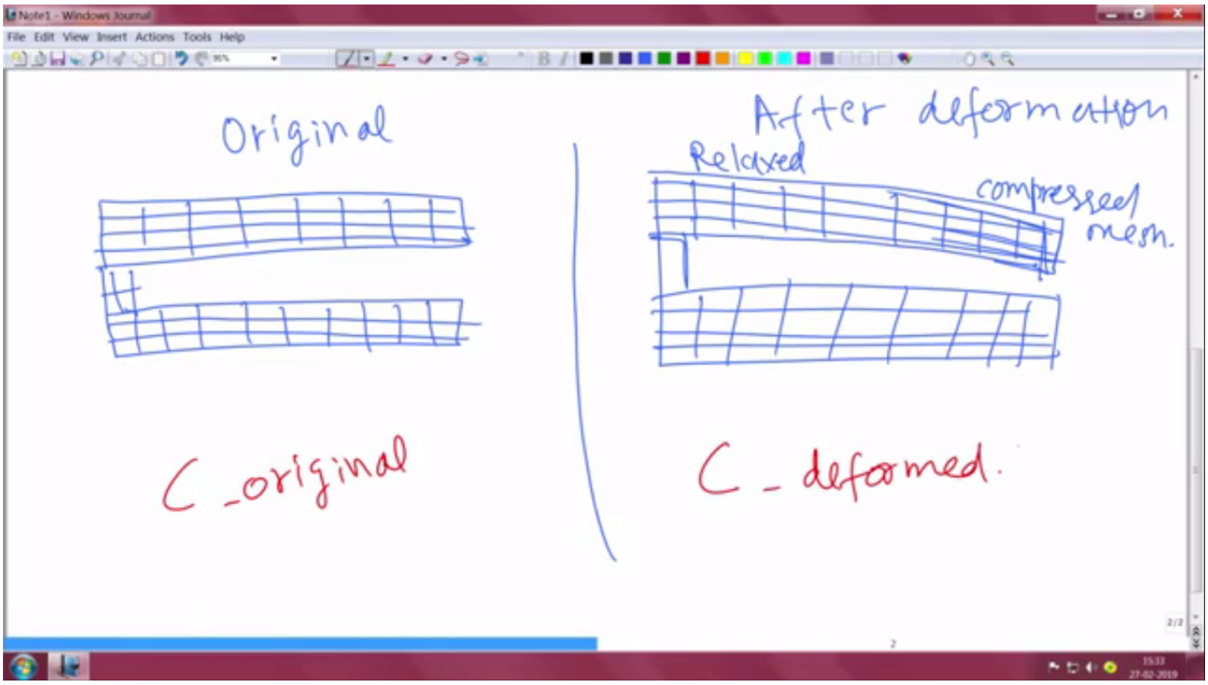
(Refer Slide Time 12:32)



So the meaning of the point that I am saying is that the mesh is going to get change and the way that I want to show you is done using the moving mesh that you can see over here, right? So the mesh that we were discussing before, that has to be taken into consideration using the moving mesh. Okay. The mesh needs to be actually changing and only then we give the potential on the changed geometry that is a deformed geometry.

So we need to work with two different geometries. The one is the original geometry that you saw in the, in the figure and then after the deformation what is going to happen once you give the electrostatic signal. So what is the capacitance of that? So you have, this is the capacitance that the original geometry that this will give and this is capacitance that the deformed geometry will give is going to change.

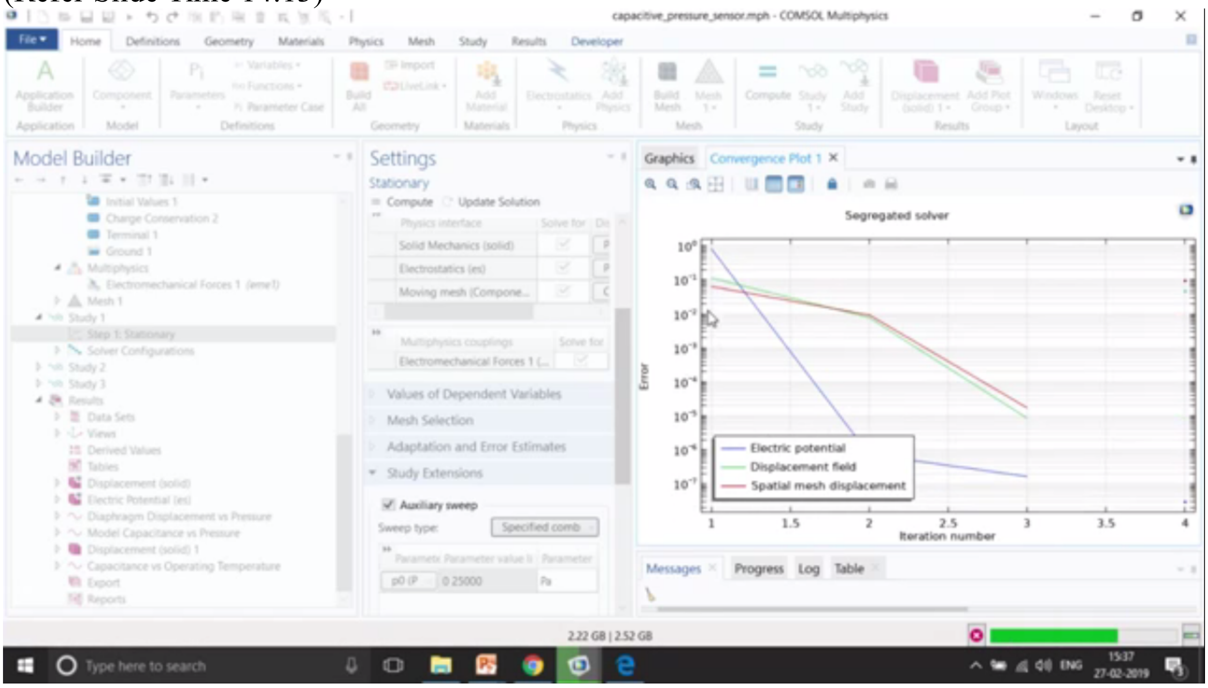
(Refer Slide Time 13:32)



So it's very important that the mesh actually change so that it will get reflected as a form of the capacitance. Okay. So let us see how does that varies.

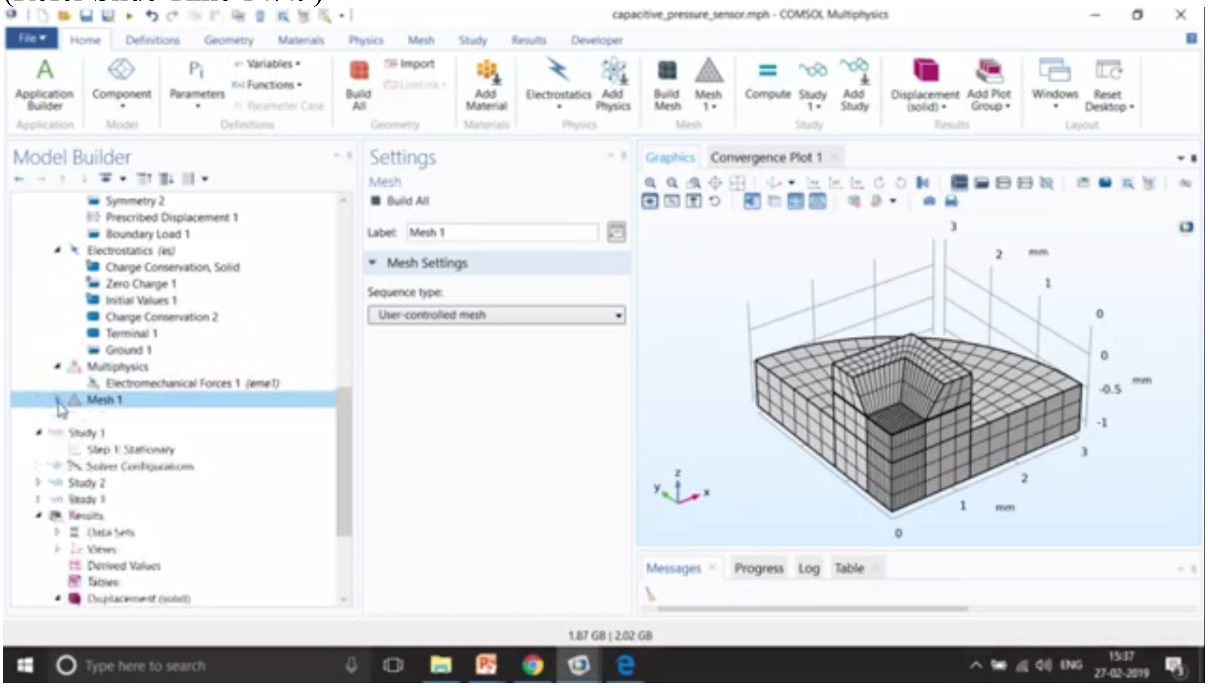
So over here they do a parameter sweep of different, different pressure. However, let us go with a very simplified way. Let us give 0 and 25,000 just to see how the pressure varies.

(Refer Slide Time 14:13)



So you can also see the convergence plot over here. Usually, the errors go below 10^{-3} to get it resolved. Okay. So we have the results. So what I wanted to showcase you are the mesh. So I would just disable this. I will first plot the mesh. So you can see that mesh is also there. If you want only mesh the, and we want to visualise the mesh. So I just make a volume mesh and this is the actual mesh, right, that is coming from the mesh over here.

(Refer Slide Time 14:49)

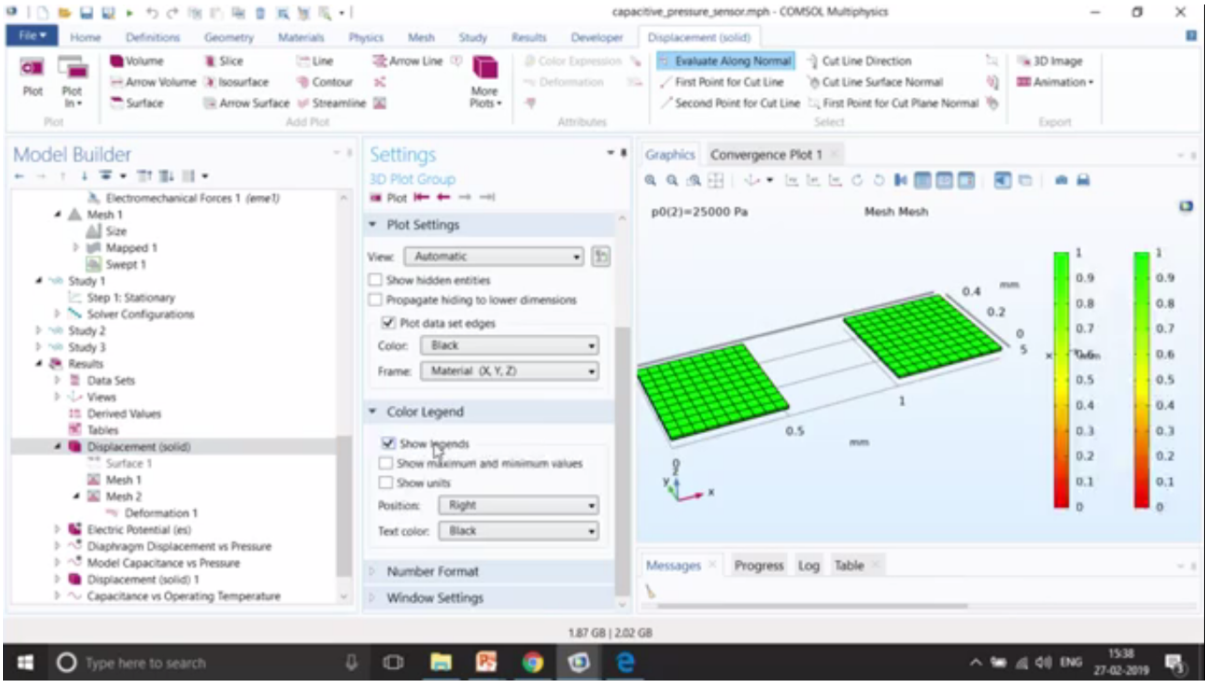


So this is the actual mesh that we have done over here that is using the map mesh. So we actually have the size node. We first mesh the bottom part and then we sweep it to the top part. Okay.

So let us go with the mesh of this particular geometry. So in the first one also we have two values. So you can see that this is for 0 and then this is for 25,000 kilo, sorry, Pascals of pressure. So you can see some deformation that was there, right? So you can see the change in the mesh and this change in the mesh is then taken into consideration while we do the simulation.

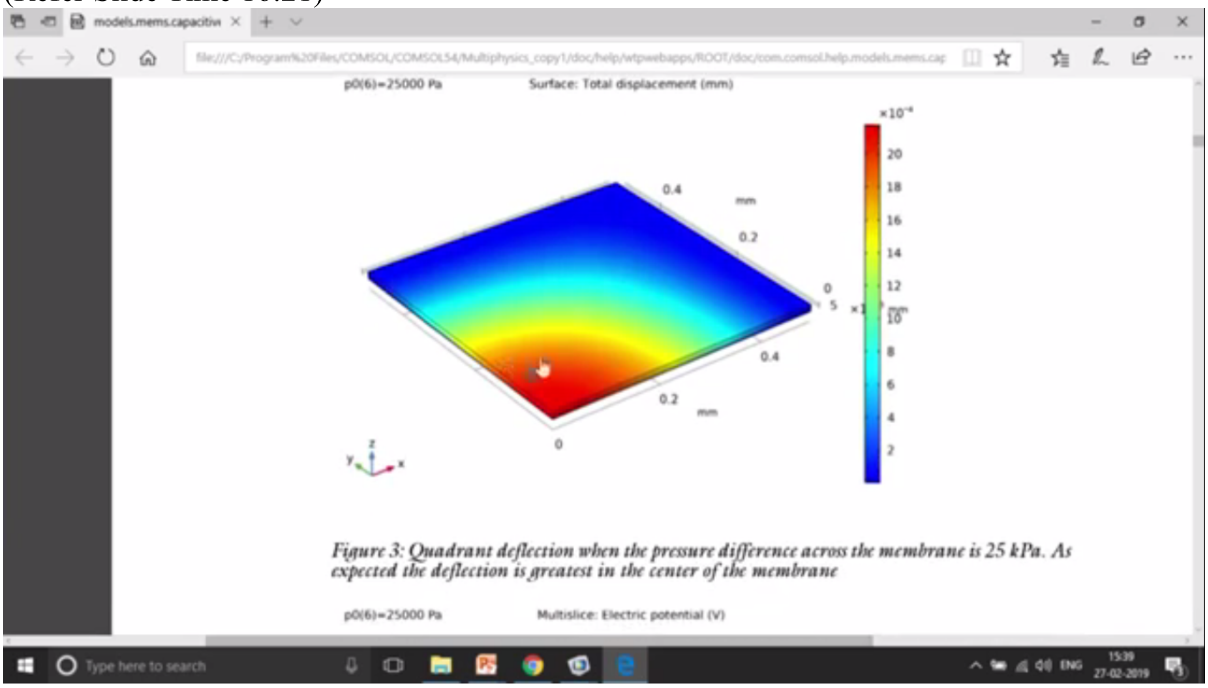
If you want to see it side by side that also is possible. Just duplicate it and have some kind of deformation to shift it towards the right side that is in order of 1 mm. So I just move it to the right side, so I can visualise both of them together. So in the mesh 2, I just write it as 25,000, right?

(Refer Slide Time 15:56)



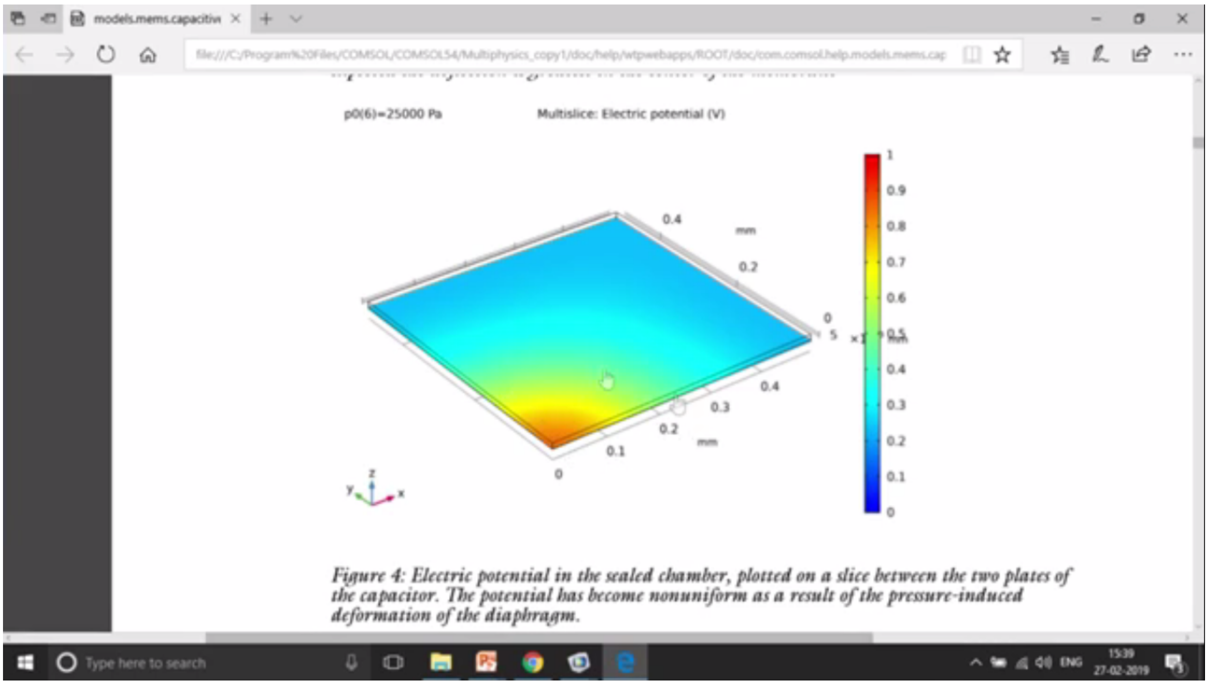
So if you want to just maximise it, so you can see that on the left side is for no deformation and then on the right side it's with the deformation. So both are having a different mesh. Okay.

(Refer Slide Time 16:21)

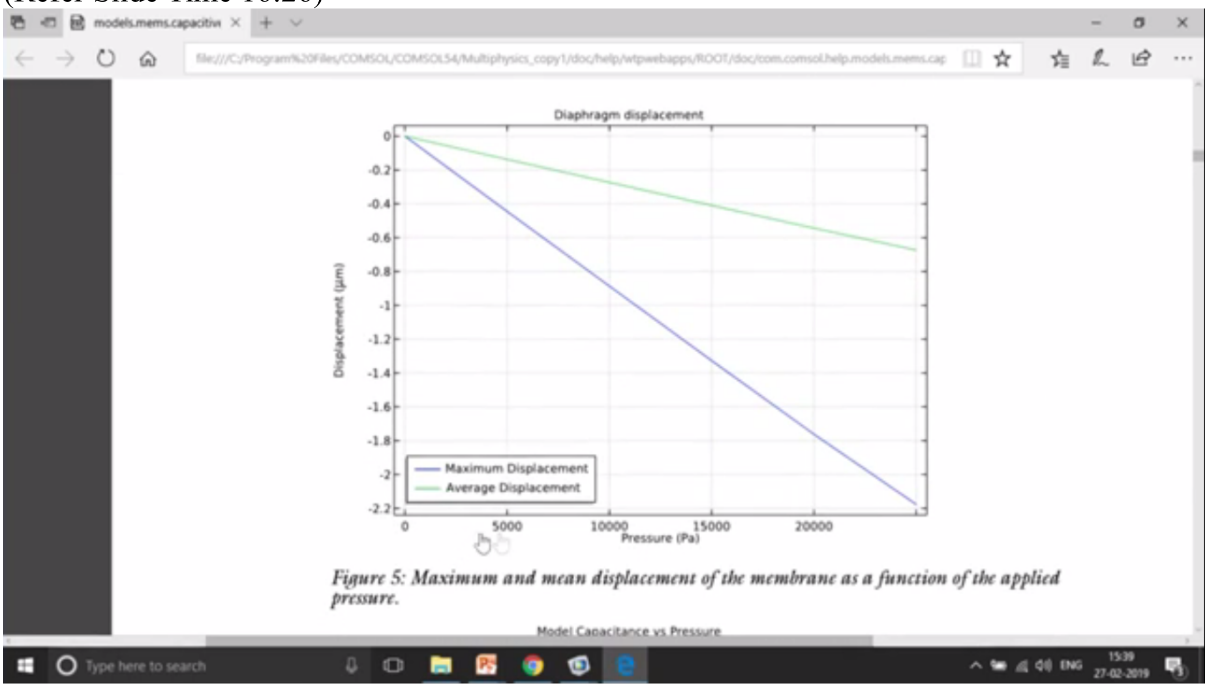


So you have the potential plot over here and then the deformation plot.

(Refer Slide Time 16:24)

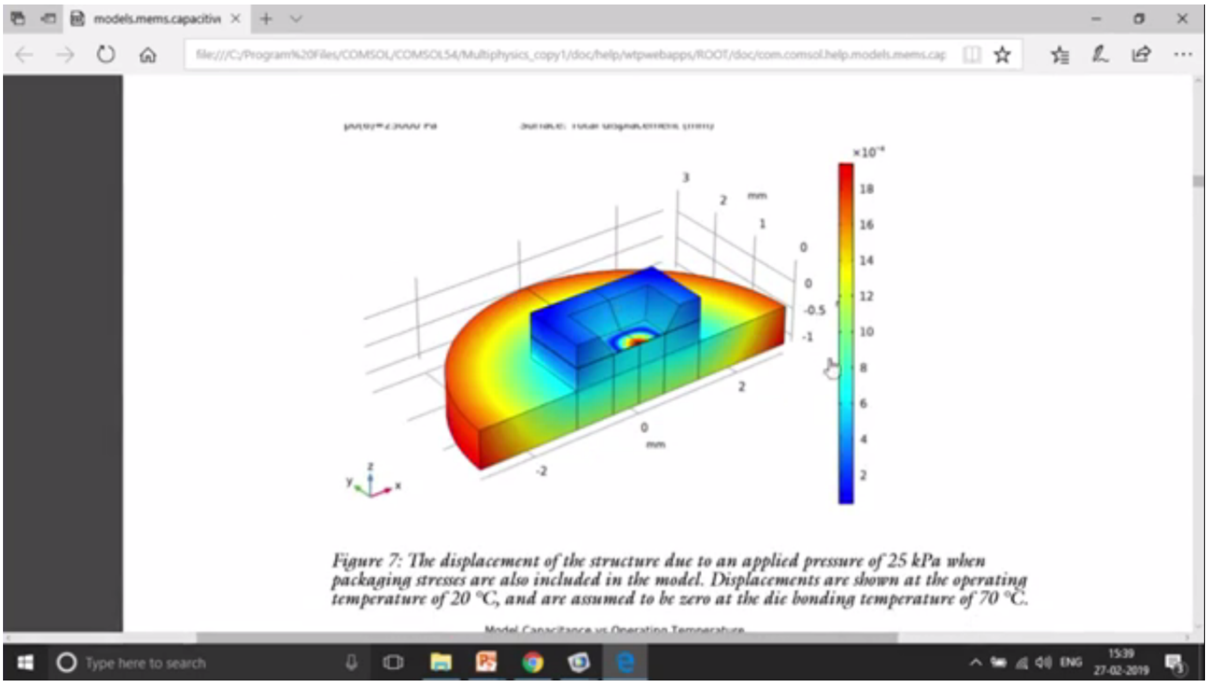


(Refer Slide Time 16:26)



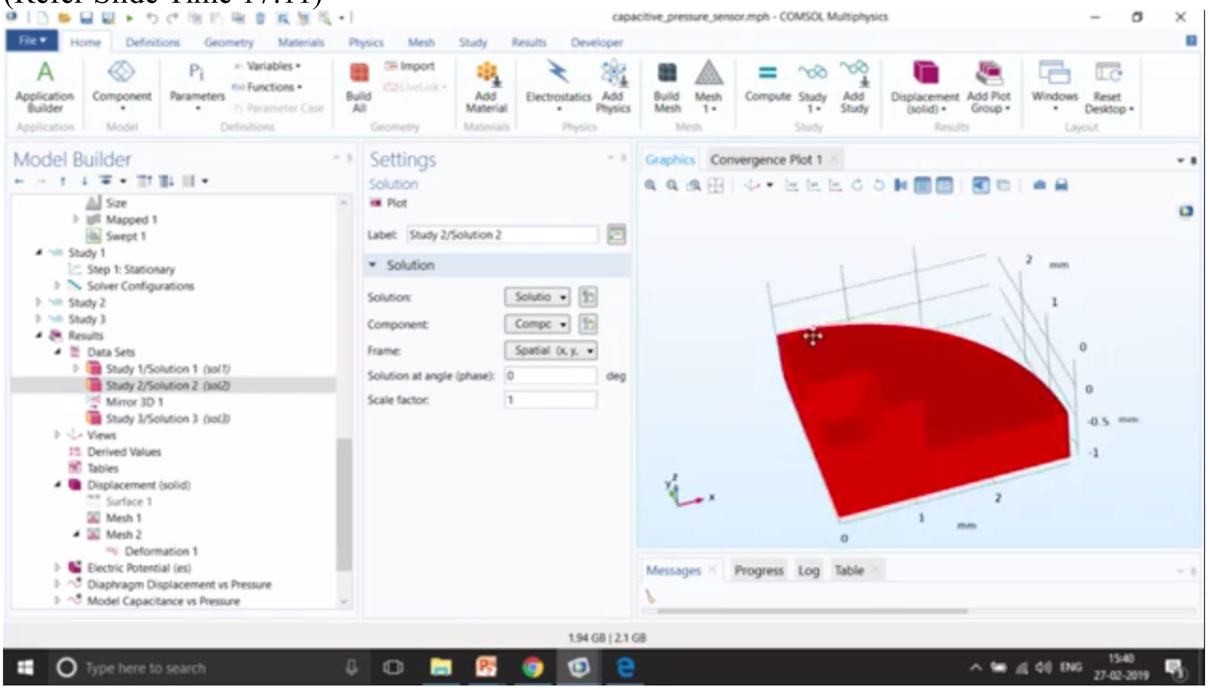
And then you can also vary the pressure from 0 to 25,000 and then see how much the change in displacement is going to occur at a particular point. Okay. You can also compare the capacitance with the analytical capacitance that is available as an equation and then see how much is it compared with this, right?

(Refer Slide Time 16:51)



So the complete simulation, so now I have done only for the quarter part, but if you want to simulate for the other part or the complete structure, you can also mirror the dataset. To mirror the dataset, you can just go to dataset and use 3D mirrors.

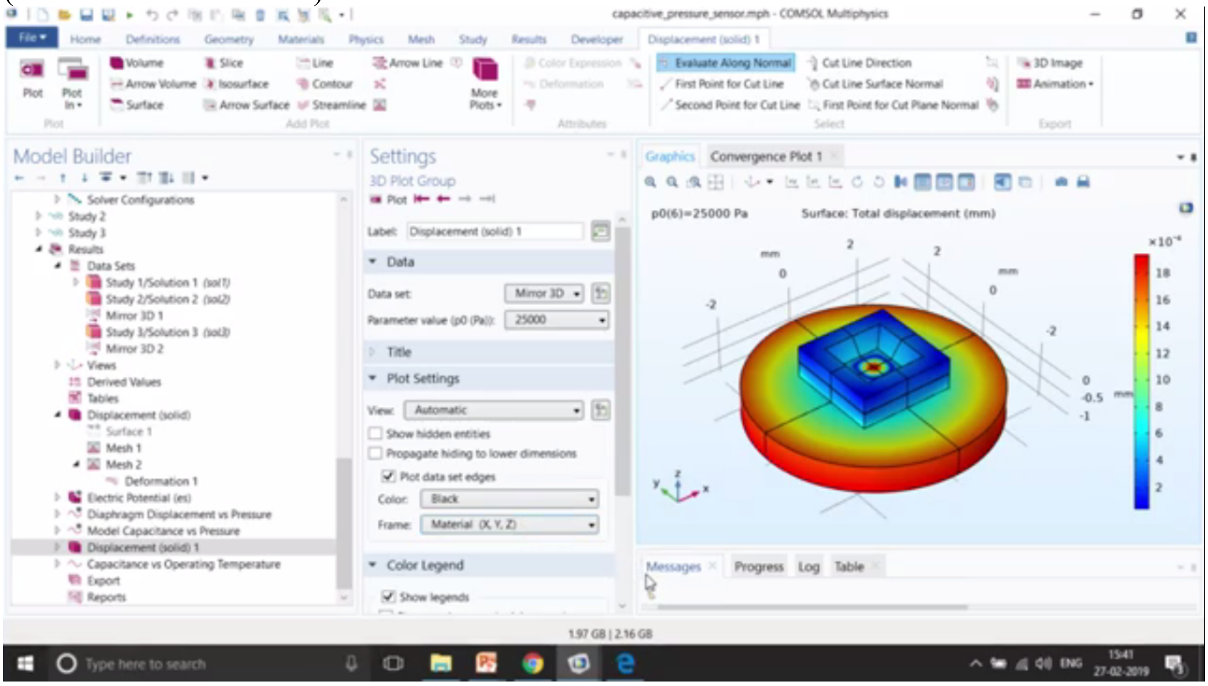
(Refer Slide Time 17:11)



So if this is my actual geometry, this is my actual geometry that you can see over here, I can use the mirror to actually make it half of the geometry. I can again use the mirror. So I can just use 1 more mirror. So let me go ahead and add one more mirror and use my study 2 study mirror, this one, and then I have to choose this zx-plane and I get the complete solution.

So I actually model only a quarter part, but I get the result for the complete simulation of the complete switch. For example, now if I want to see the complete switch, I can just change it to my mirror 3D 2 dataset that is nothing but this dataset. Just use this one and just plot it and I get the result for the complete structure where I have only modelled for a quarter of the structure.

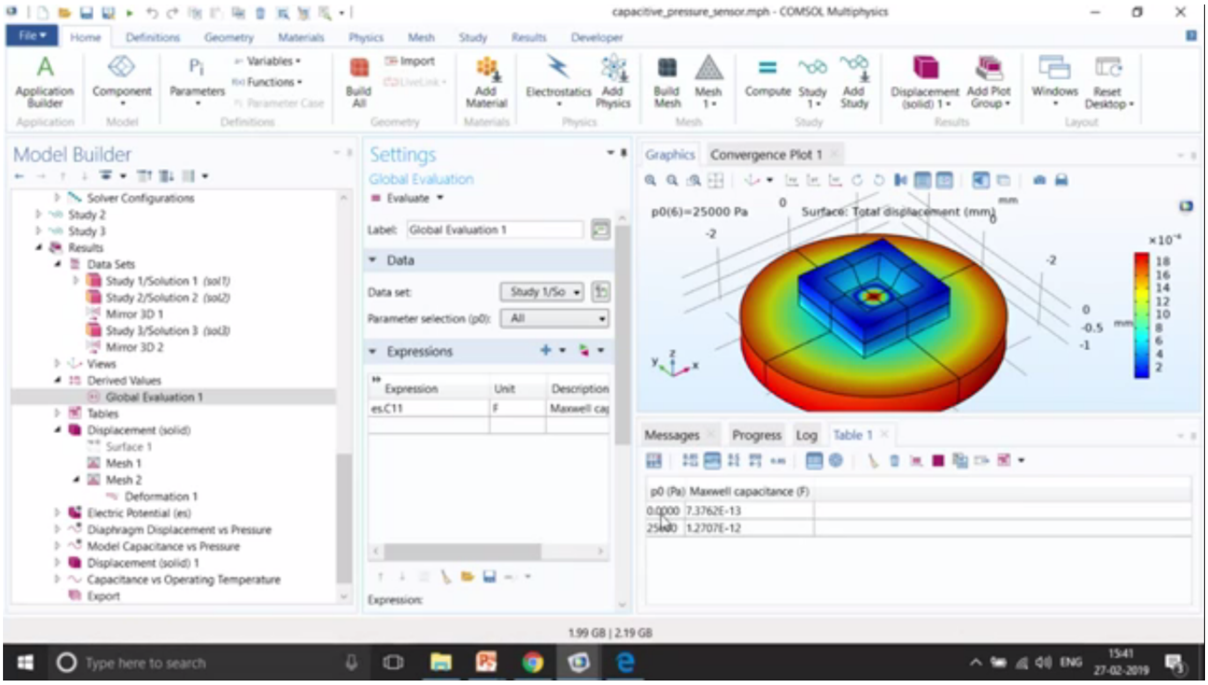
(Refer Slide Time 18:08)



Okay. So this is how the pressure sensor work. You give a particular load which changes the geometry of your structure that means the meshing in the case of numerical software such as COMSOL and then on that changed geometry we give electro, electrical physics and from there we understand what is the capacitance.

The way to evaluate capacitance is very simple. Just right click on the Derived Values, Global Evaluation and just search for capacitance. You can see that you can get the capacitance with the value of esC11. So we solve for two pressures.

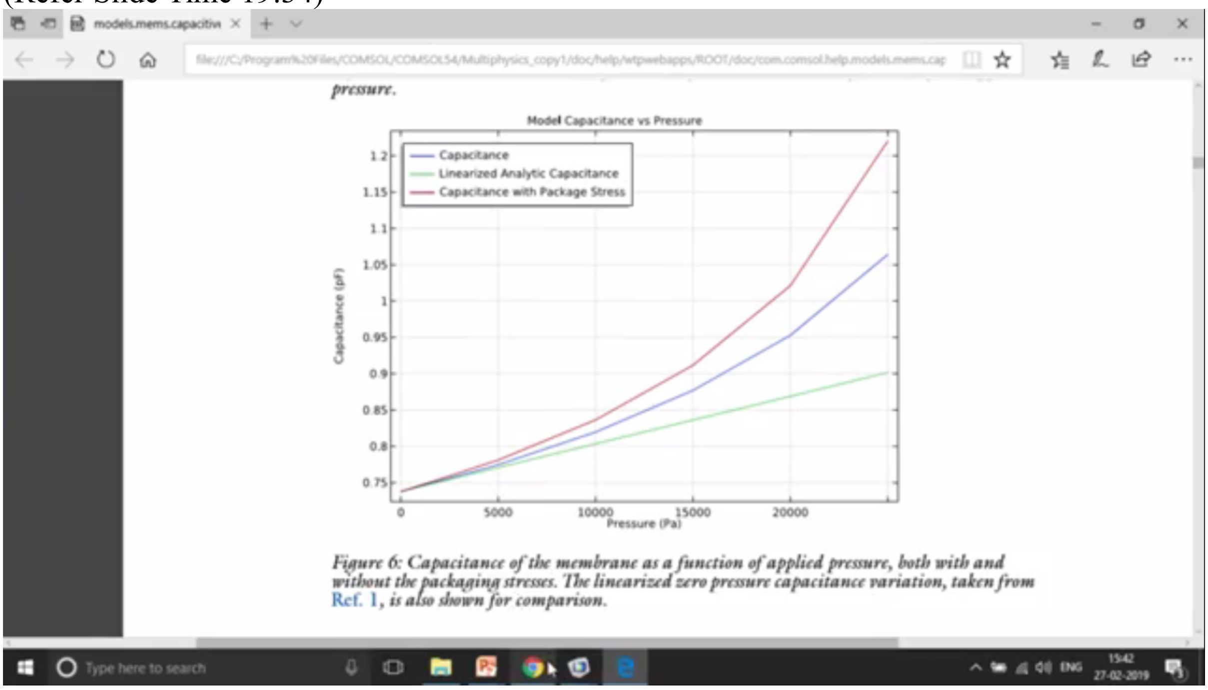
(Refer Slide Time 18:56)



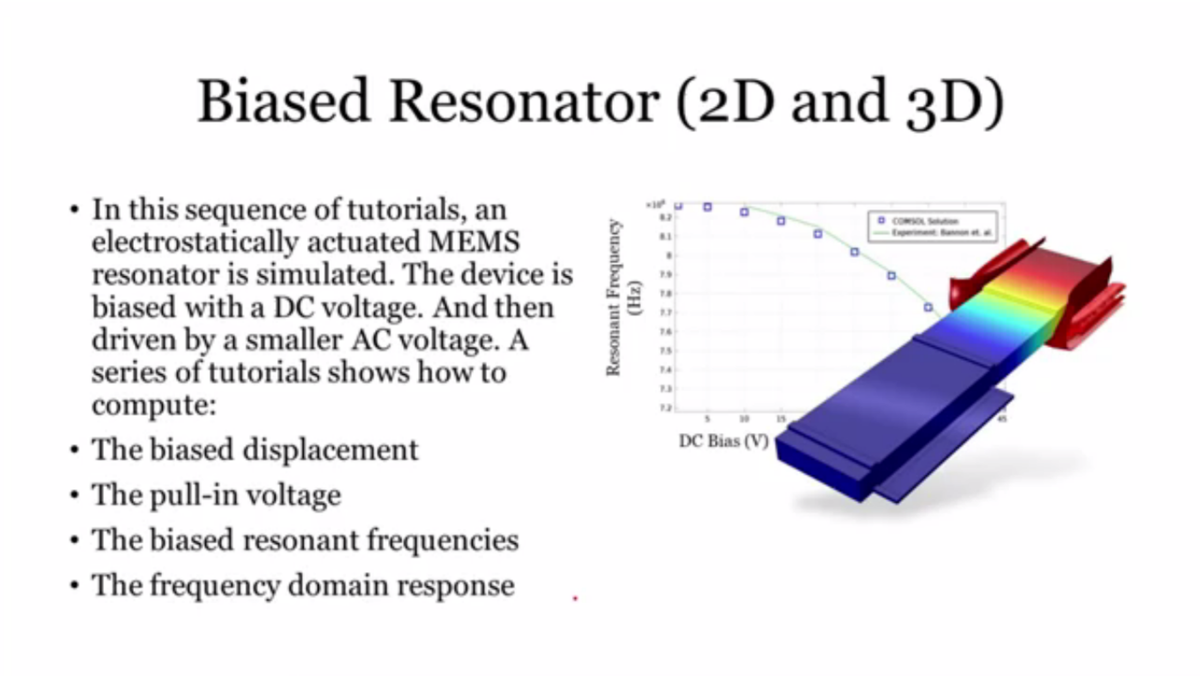
So as you can see over here, the capacitance for two pressure is changing. I can make it as a unit of picofarads and then again evaluate. So as, as expected the d value is going to that the distance between the two plates is reduced because of more pressure and it is inversely proportional to the capacitance. That's why the capacitance is increasing as the pressure is increased.

You can also go to have a line graph of how it represents the change in capacitance based upon the pressure as you can see over here.

(Refer Slide Time 19:34)



(Refer Slide Time 19:40)



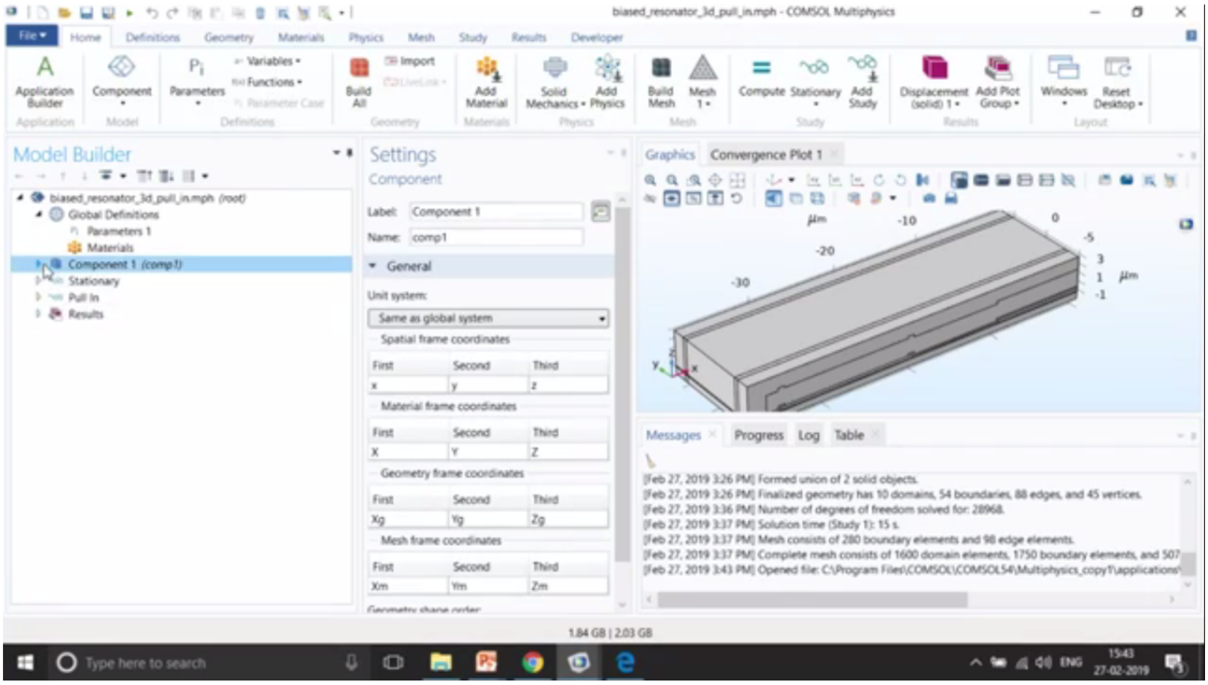
The next example is biased resonator. This is an actuator beam kind of an example where we give a particular pull-in voltage and because of that pull-in voltage how much deformation or displacement or actuation of that cantilever beam is going to take place has been modelled.

So if you want to see that particular model, you can just go over here. I think it's available in MEMS module somewhere. So let me just go through. Yeah. In the MEMS module, in the actuators, yeah. So we have made different kinds of analysis on biased resonators. So you can have particular frequency models where you want to know what kind of modes with which it is going to fluctuate.

We have pull-in of this voltage, I mean, set a particular voltage where it’s going to touch the bottom plate kind of thing analysis. Then we have 3D structures again, right? So you can actually open any of that model. First, go to the documentation and then open the model. Okay.

So I will just open that particular model, one of the model and explain it.

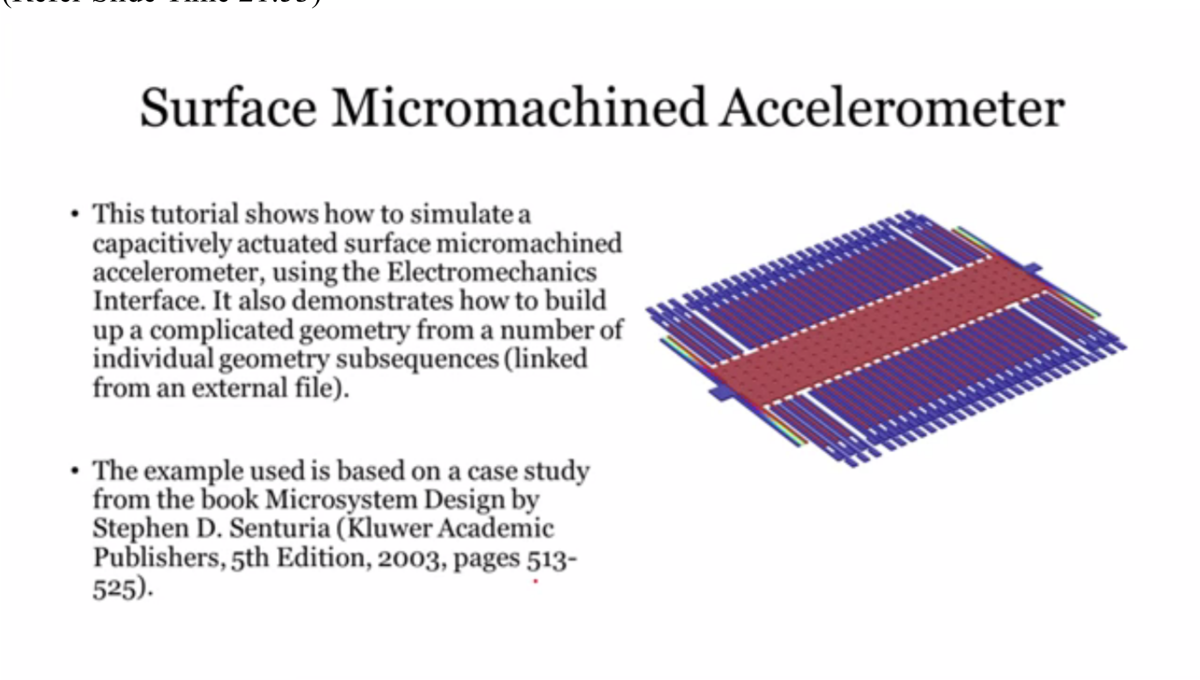
(Refer Slide Time 20:51)



Again, the geometry part is been important. Then the material part. So here we have used polycrystalline silicon over here. So if I enable the wireframe rendering, it's polycrystalline silicon that you can see over here and then we have silicon nitride in the bottom part and silicon oxide over here and the rest of the domain is air domain. Okay.

In structural mechanics or solid mechanics, we give a particular fixed constrain in the bottom part and symmetry on this part and then on electrostatics we give a particular voltage and then ground on the bottom and terminal on this particular boundary. This is little bit more complex. To start with, I will suggest you to go to the first example of 2D resonators. Okay.

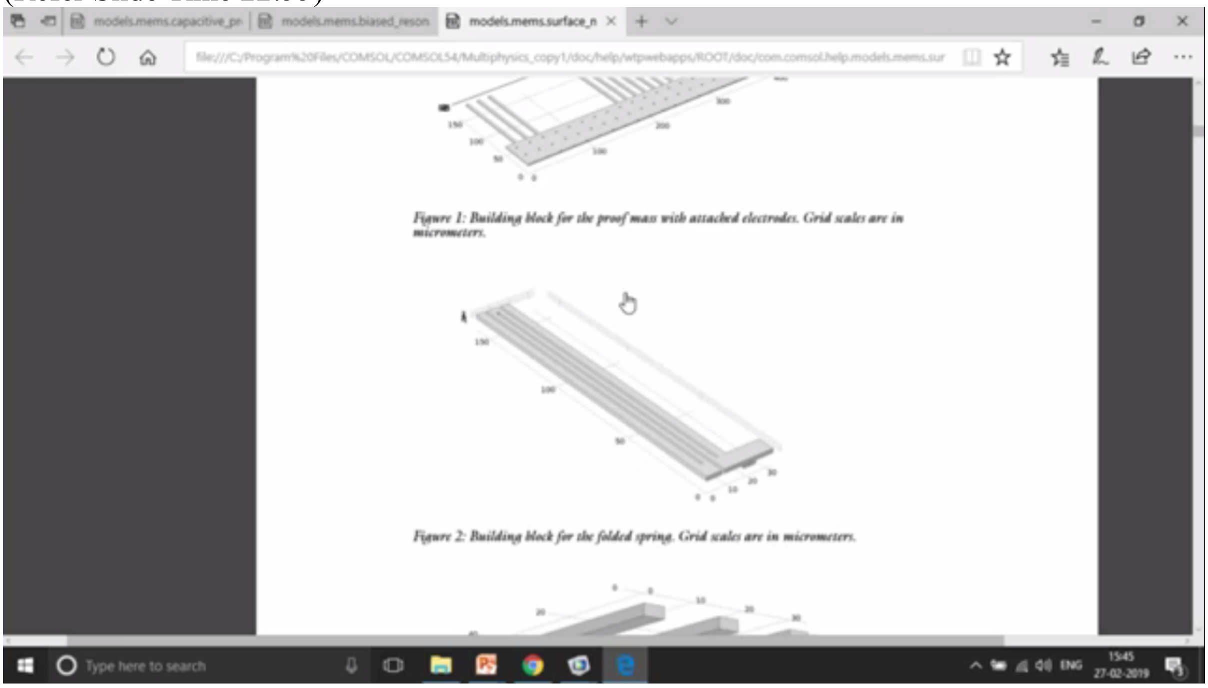
(Refer Slide Time 21:53)



So Let us go ahead and this is an example model of an accelerometer. So this is also available in the Application Library of COMSOL. So if you want, you can just go. So this kind of devices are usually used to understand the acceleration of a particular device, maybe your mobile phones, even, so there are many games where which accelerometer devices have been used in cars, in vehicles, in aircrafts, accelerometers have been used a lot and now they are also moving from accelerometer to SATNAC (inaudible), so which actually works on the basis of optics, but in this case it's purely (inaudible) mechanical approach.

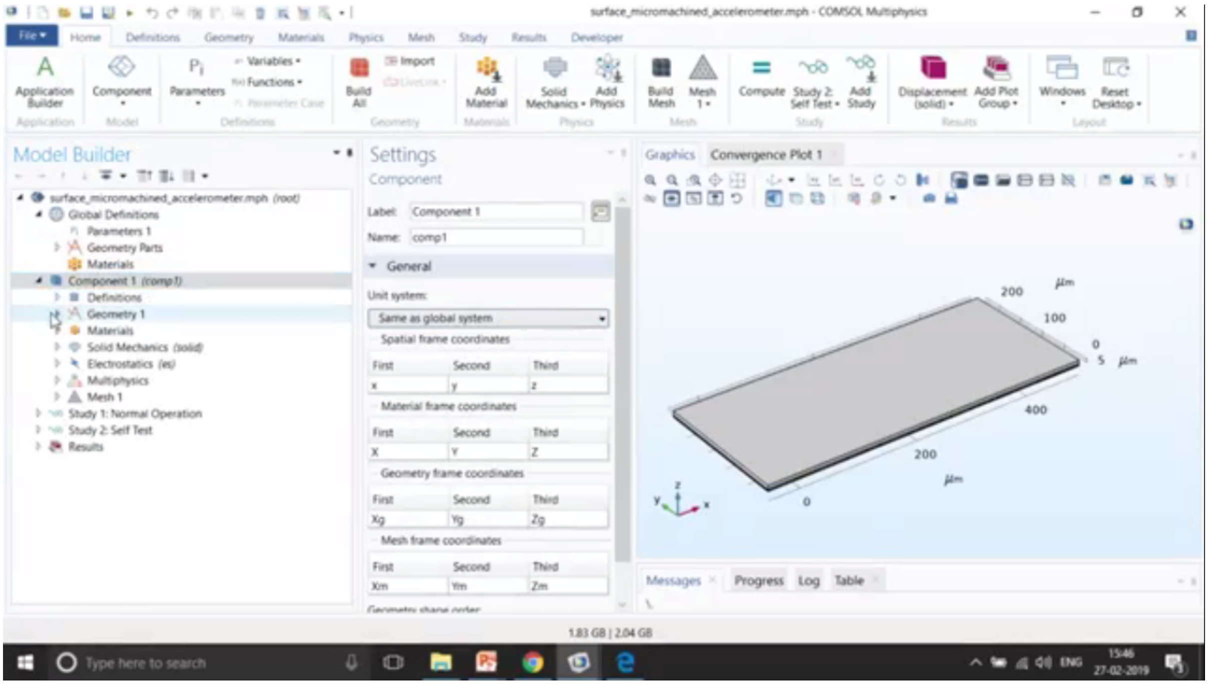
So let me just go ahead and open this particular model. I have this model over here somewhere. Yeah. So we have an example of surface micromachined accelerometer the geometry of which is little bit complex.

(Refer Slide Time 22:55)



So we have the proof mass, the folded spring, the building blocks that is the electrode arrays, both the electrode arrays and then the analysis part. So let me just open this model.

(Refer Slide Time 23:10)

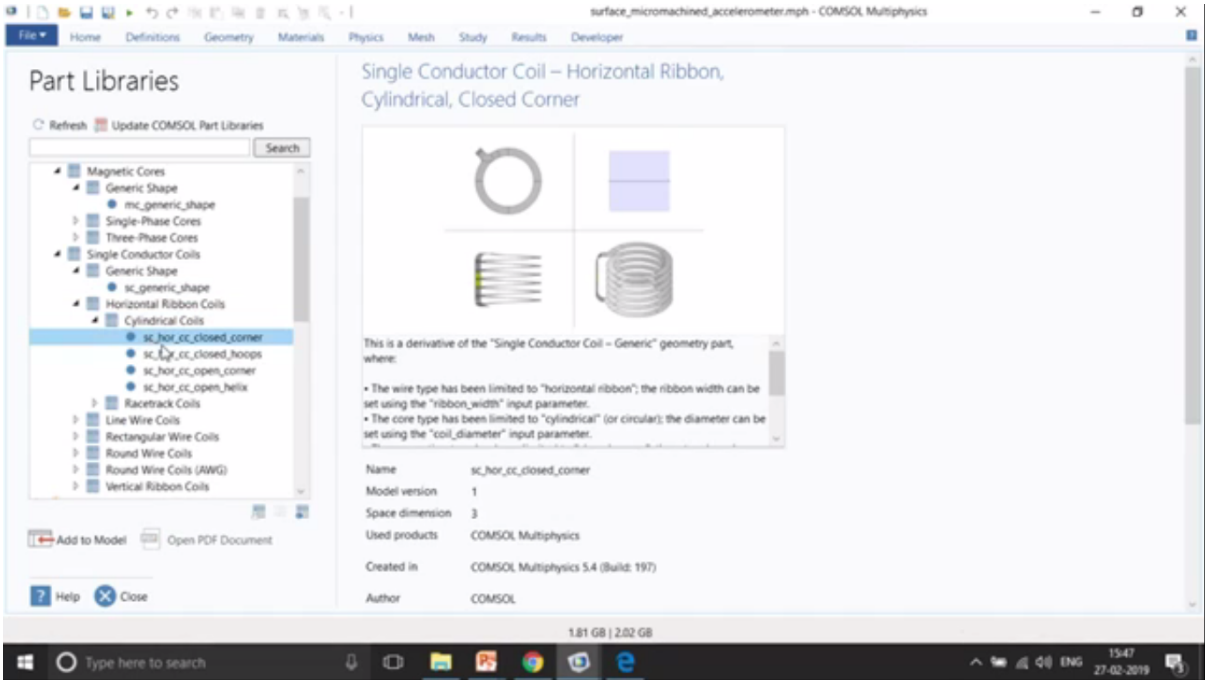


So this is the structure that you can see over here. Geometry part has been modelled over here. So it has been introduced from the geometric parts.

So there are many structure that are already available in COMSOL. So if you want to understand what are those materials, you can just right click on the Global Definition and go to Geometric Parts and go to Part Libraries. So if you go to Part Libraries, there are many structures that are already available.

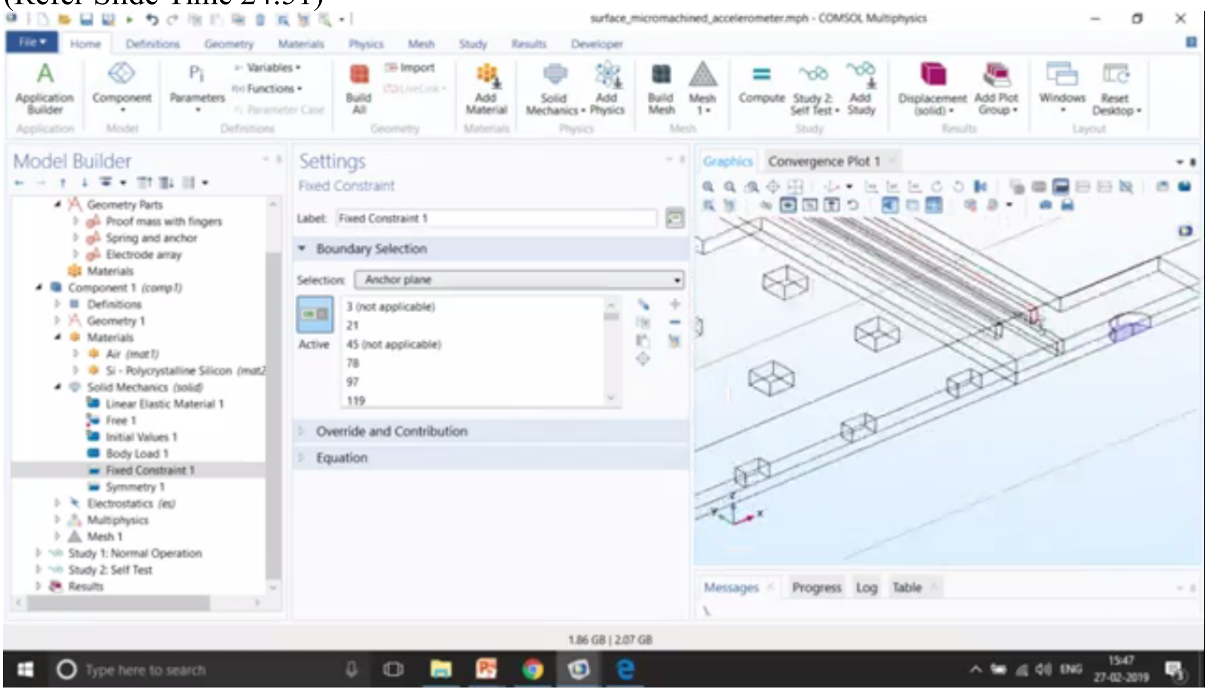
For example, you are looking for lens, a different some kind of a lens. Then you can actually have the lens introduced from here directly from geometry and based upon the focal point, the complete design of lens is going to change. If you are talking about some kind of beams and bolts, you can also look, try to install from here. Okay. You have codes, magnetic codes, different kinds of codes, magnetic, that is also possible. Different kinds of coils are also possible as you can see over here.

(Refer Slide Time 24:09)



So over here the model has introduced these kind of domains from the Part Libraries directly and from there they have intro, that have called in this particular interface and then applied the material properties. There is air and polycrystalline in the bottom and then finally the physics part, that is the solid mechanics part that talks about a particular fixed constant. So we give a fixed constant at the bolts that you can see over here, the blue part over here, right?

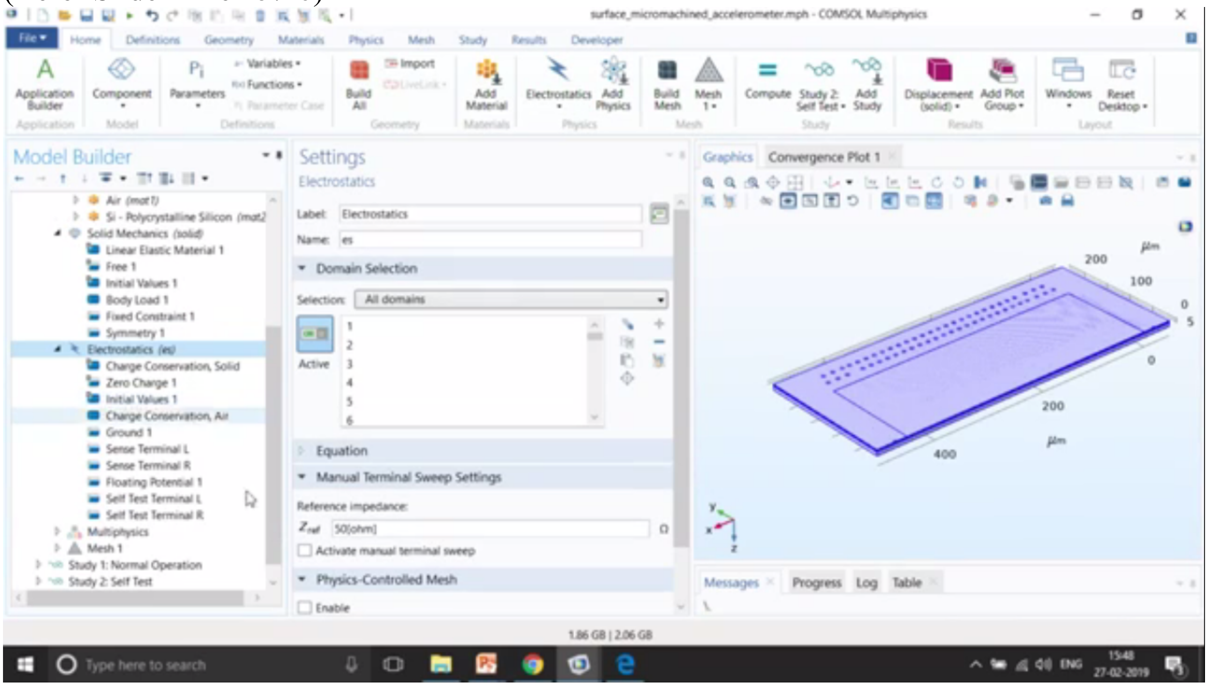
(Refer Slide Time 24:51)



So these are the different kinds of fixed constant. Here there are many. You can see over here. So all, all of them are bolts. That's why fixed constants have been given. The blue part signifies the fixed constant.

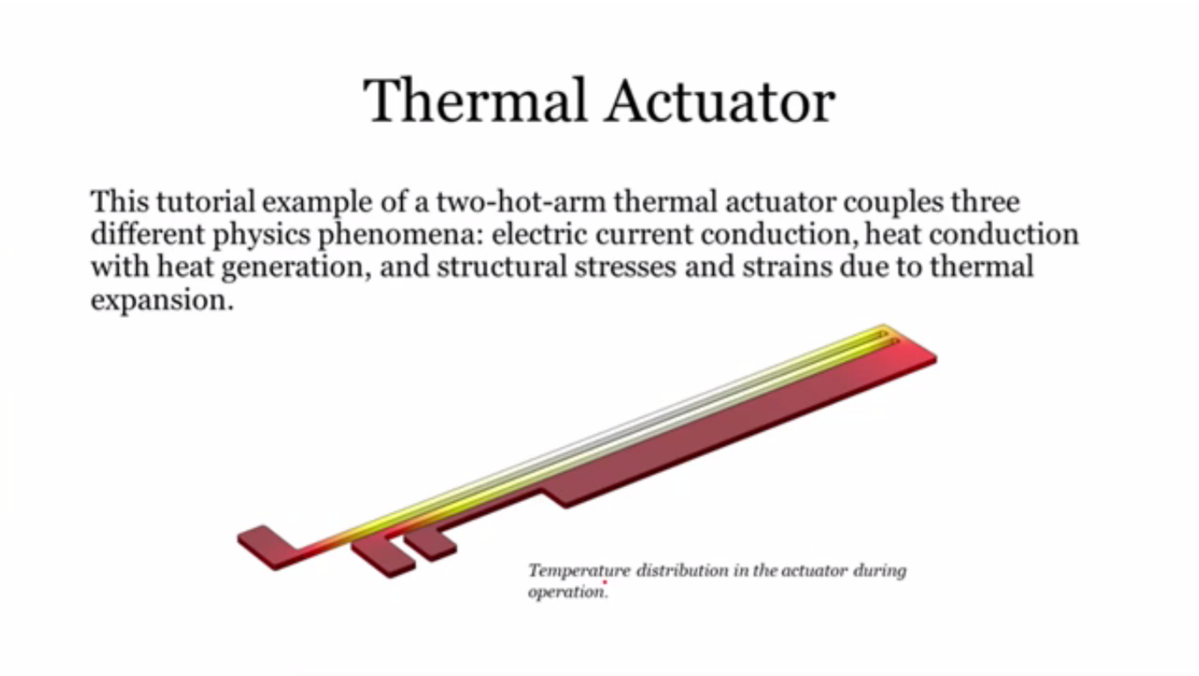
Then the symmetry part because we have modelled only half of the accelerometer, not the complete accelerometer. That's why we have given you a symmetry boundary condition over here. So it will in the post-processing we can actually reflect what is on the left side to the right side with the same results.

(Refer Slide Time 25:26)



Then the electrostatics part. Then we have sense terminal and floating potential as we discussed before to withdraw a particular voltage. Okay. Then we have Multiphysics coupling with couples the solid mechanics with the electrostatics, and then we have two types of study. Both are stationery, but, however, first is normal operation and second is for self-testing the details of which you will find it in the documentation.

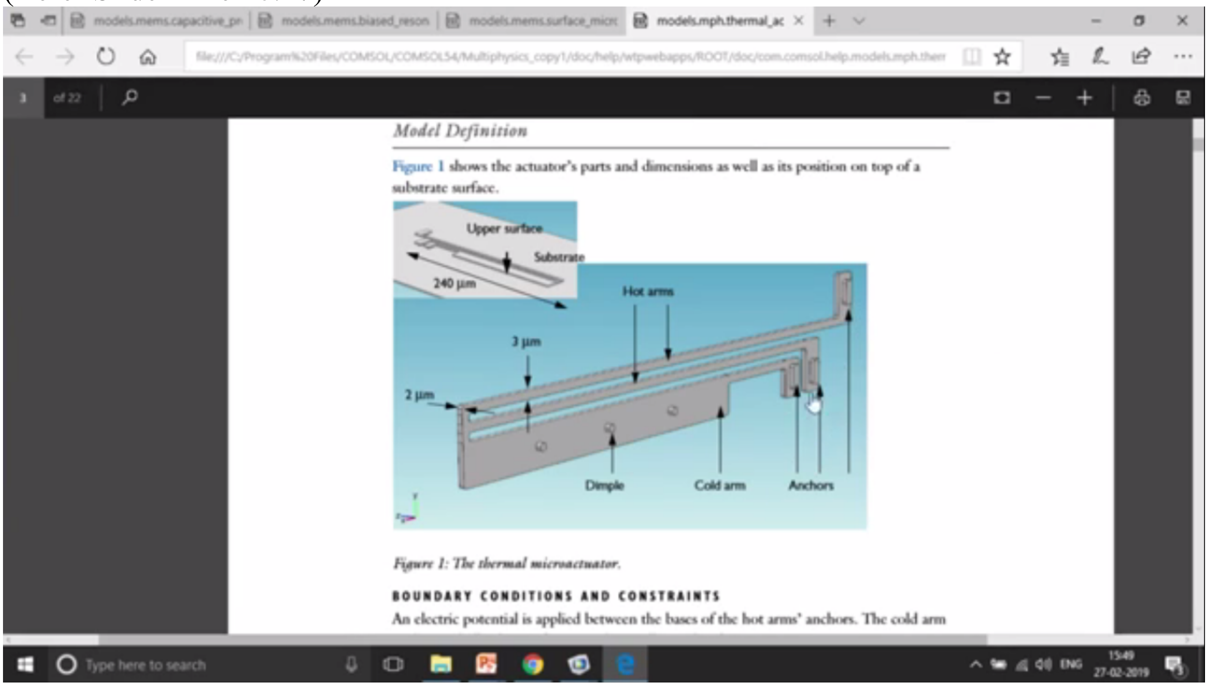
(Refer Slide Time 25:54)



Okay. So this is the last example for the actuator is thermal actuator. In this it works on the principle of thermal expansion. So let me just go and open this particular example. So I just search for thermal actuator and you can see over here there are two types of thermal actuators and along with a particular tutorial model also.

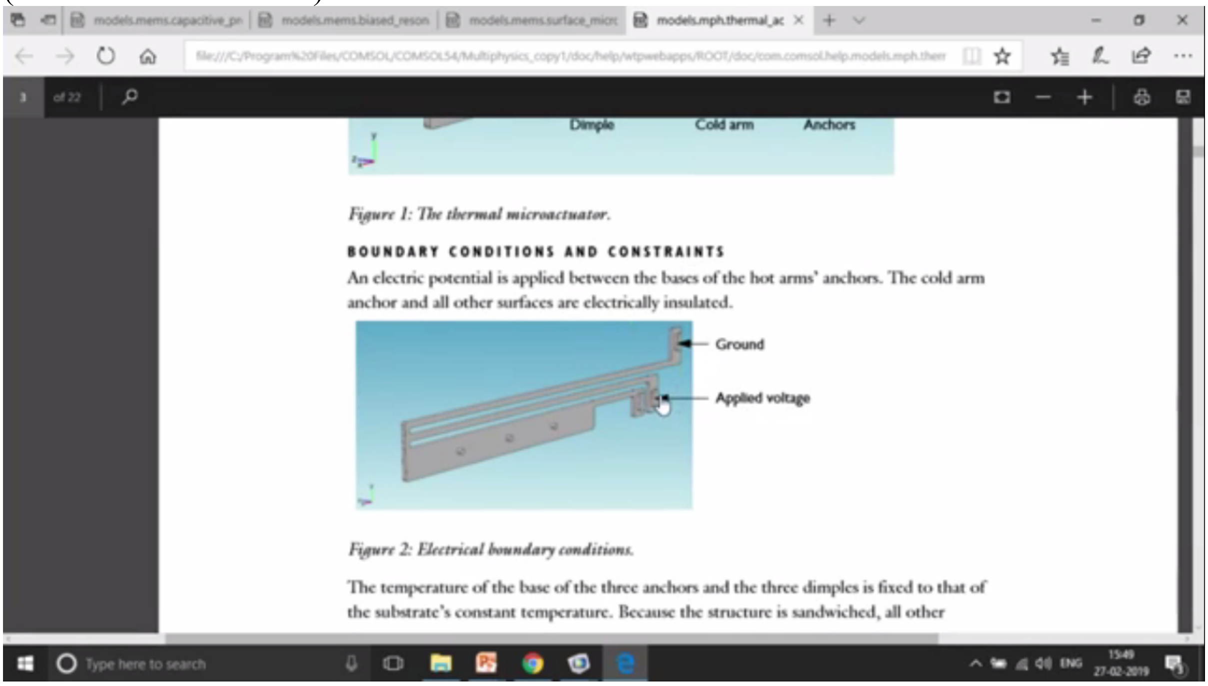
So let me just open this documentation. This is what demos that we will do in this session.

(Refer Slide Time 26:27)



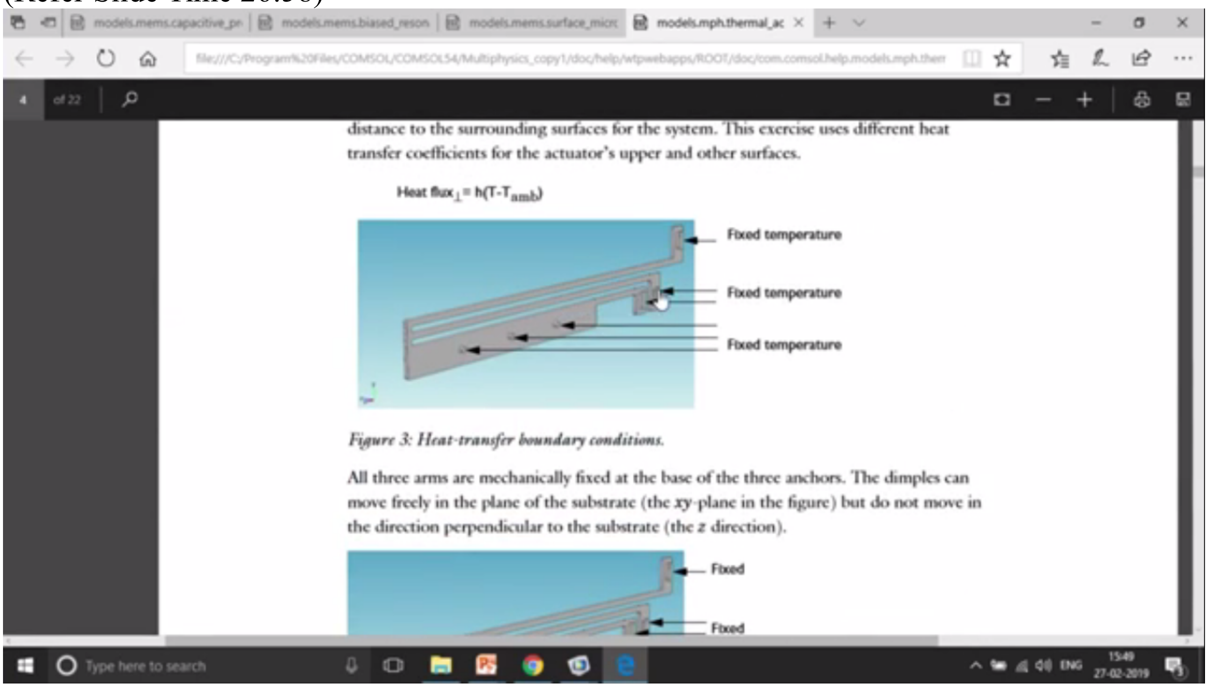
This is a thermal actuator that you can see over here and these are the anchors. This is the cold arm. The dimples are there with which it's going to get attached with. These are the main two hot arms are where the temperature would be rising because of Joule heating and these are the boundary conditions.

(Refer Slide Time 26:45)



We give a particular voltage and ground it on the other side. We also give some temperature boundary condition that is fixed temperature on these boundaries and this boundaries and fixed temp at the dimples that you can see over here.

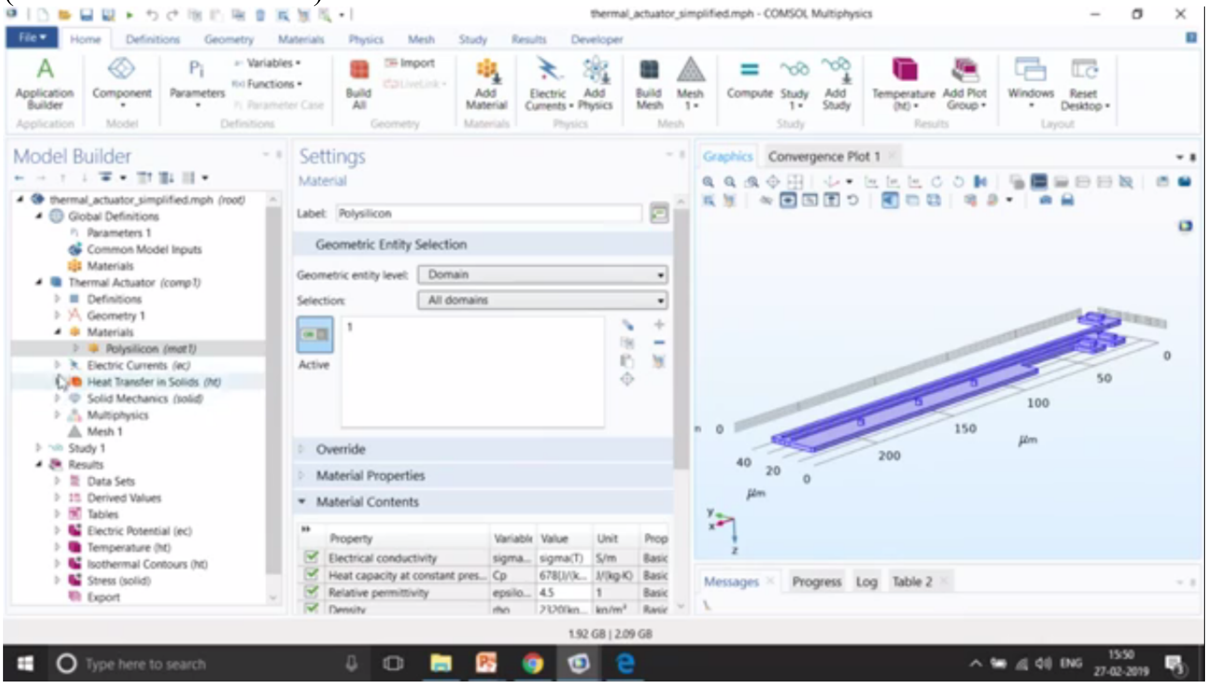
(Refer Slide Time 26:58)



And then we give roller boundary condition so that it can actually roll while it's moving along laterally like up and down in this case.

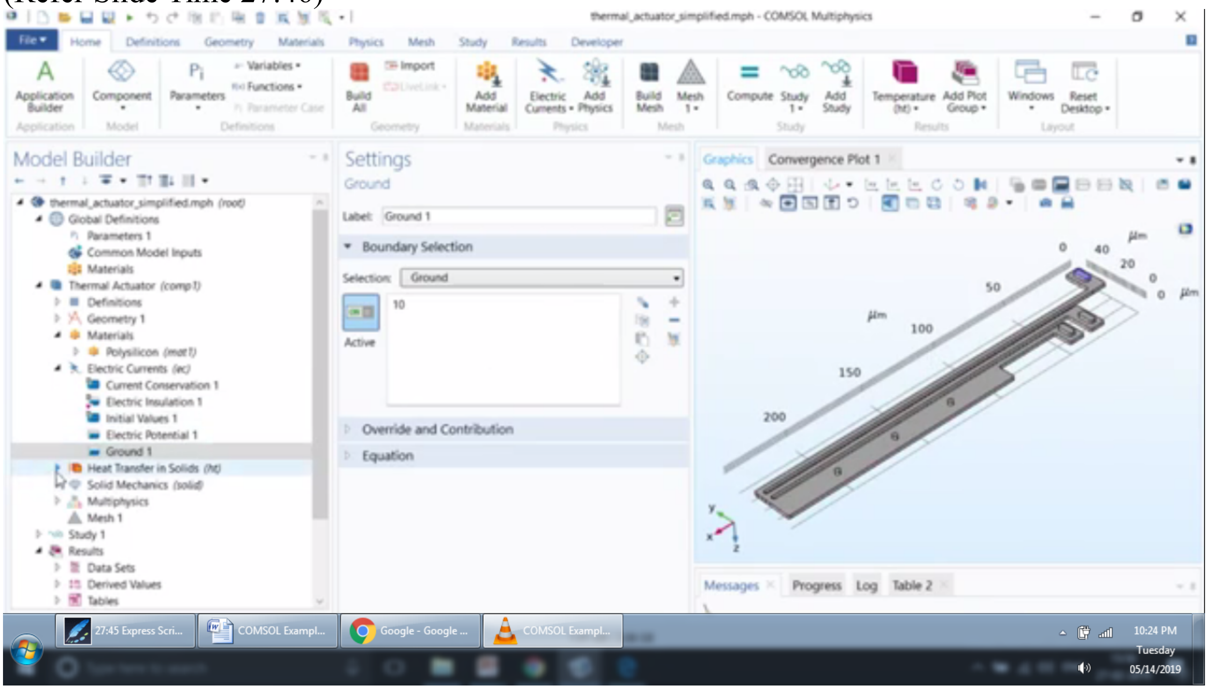
Okay. So just open this model. We are also going to do a demo on this model. That would be after the session and here you can see it’s made of polysilicon, right?

(Refer Slide Time 27:22)



And then we give, so here there are three physics. Earlier we used to work with only two physics. Now it's addition of the heat transfer physics also to take into account the temperature rise and because of the temperature rise, the thermal expansion. So this is the electric current interface with which we give a particular voltage in this particular boundary, okay, and then ground at this particular boundary. Okay.

(Refer Slide Time 27:46)



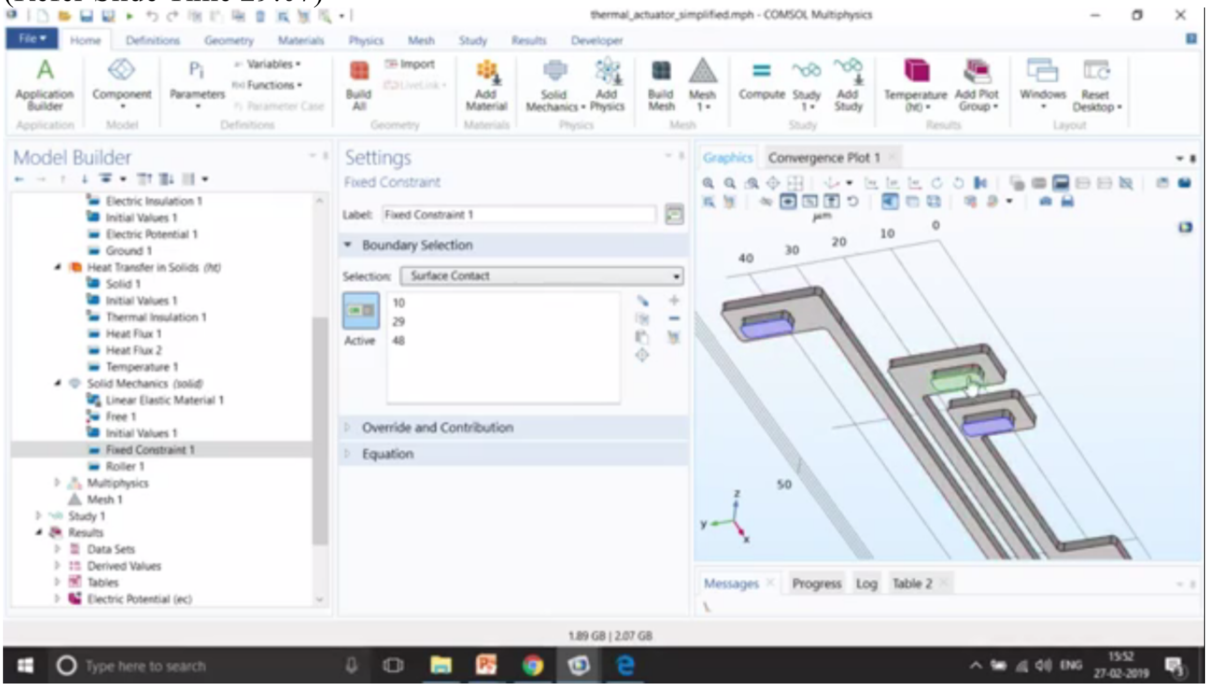
And then in the heat transfer we give temperature boundary condition that is the room temperature at the, these points and then we give heat flux. So it will be also getting cooled with a particular conviction, so convictive cooling. So that is defined as HTS, HTC_S. This

HTC_S had been defined somewhere over here as .04 W per millikelvin over here, right? So this is heat transfer co-efficient.

So that is the upper part and the bottom part is also getting cooled up with a different heat transfer co-efficient that is HTC_US where HTC_US has been defined over here. So that's divided by 2. Okay. Let me just open it. Yeah. So it's 0.04, the upper part, .04/2 micrometers and the bottom part is 0.04/100 micrometers so the top part is more as is getting cooled quickly, quickly as compared to the bottom part.

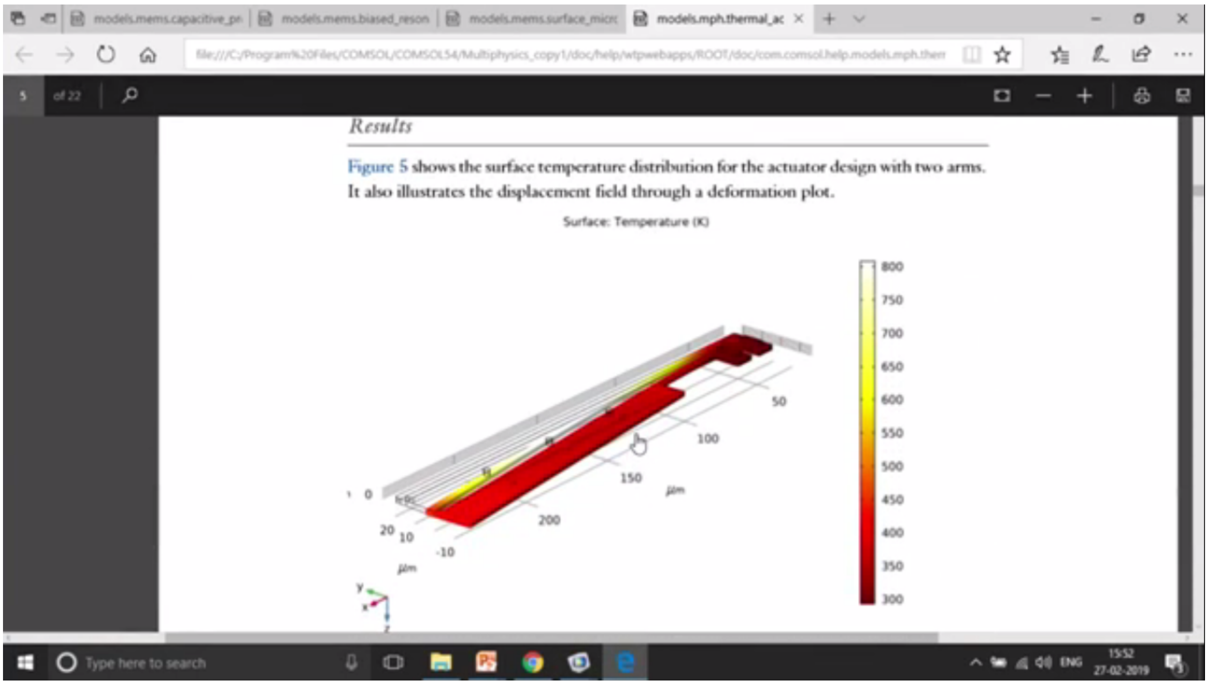
The last part is the solid mechanics part. Of course, we are going to give some kind of fixed constant. It's even somewhere over here as you can see over here in the bottom.

(Refer Slide Time 29:07)



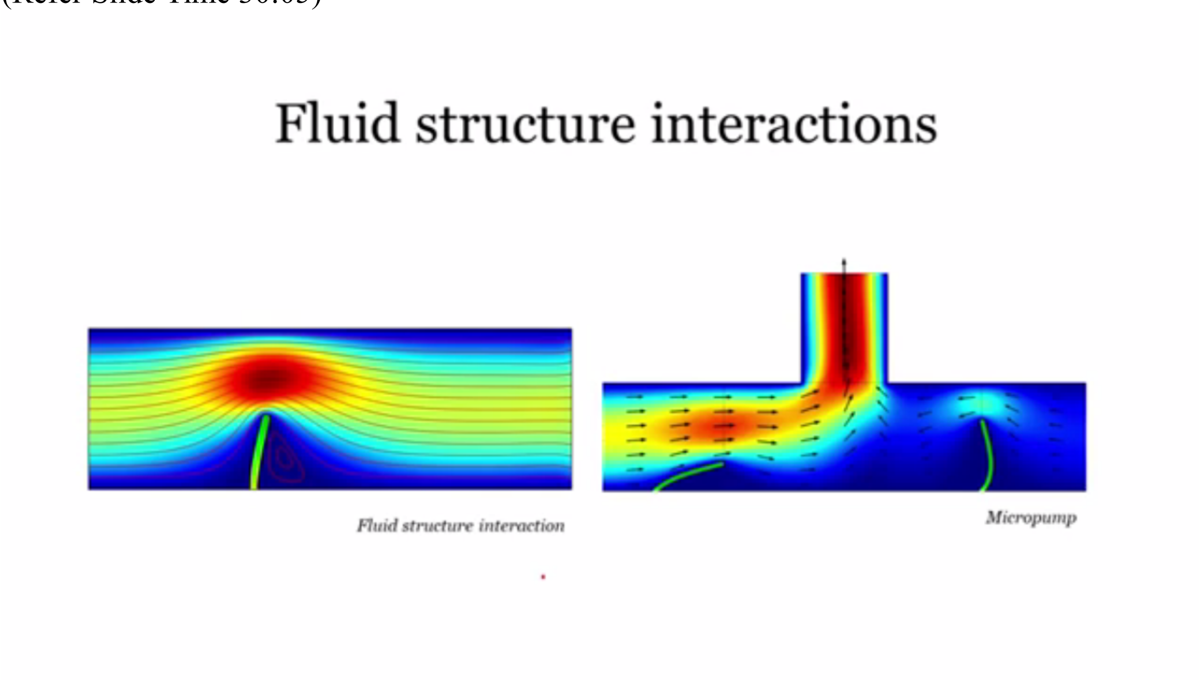
This is the three places where fixed constants have been given and then the electromagnetic heating is taking into consideration. That is this exactly couples the electric currents with the heat transfer in solids. Okay. And you can just go to study and get the results. The results actually looks like this.

(Refer Slide Time 29:28)



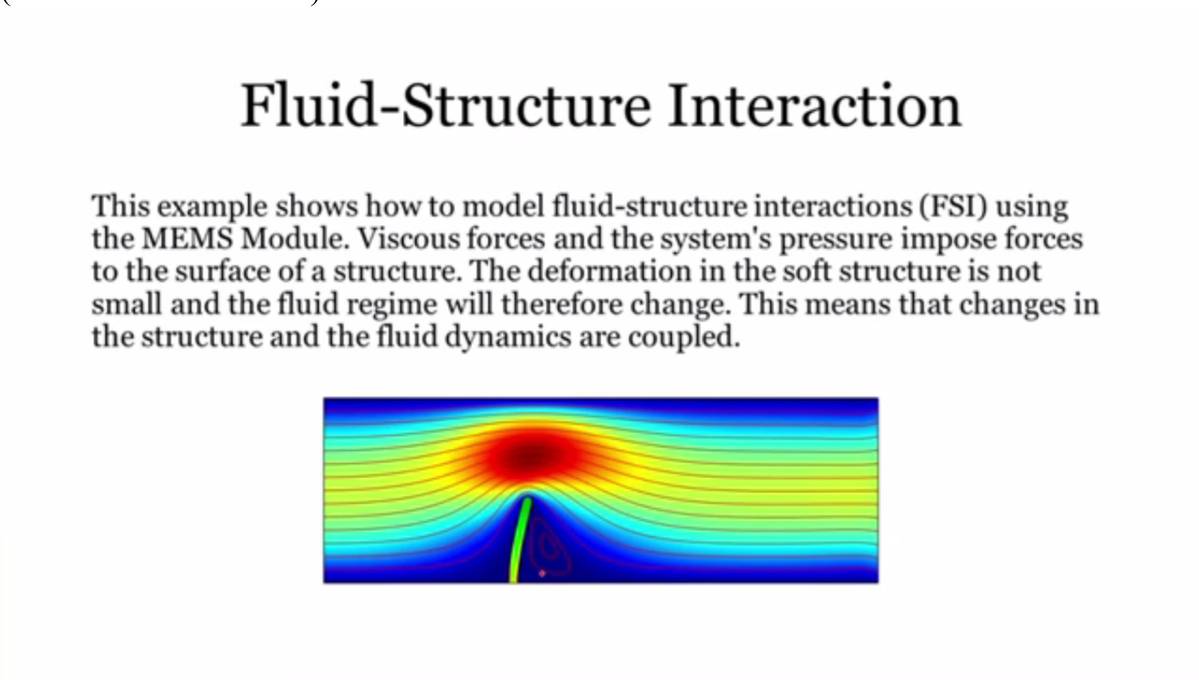
So you will see that the deformation is actually going to takes place like this. We have given a roller boundary condition. That is why it cannot move up and down, but it is allowed to move in Y direction, right? It is not allowed to move in Z direction, but it is allowed to move in Y direction. That is laterally it can move, right? This is the actual geometry that you can see in the black lines and the deformed geometry that you can see is at the red structure that you can see. Okay.

(Refer Slide Time 30:05)



The next part is the fluid-structure interaction. So we talked about thermal. We talked about structural. We also talked about electrical a lot, but what about coupling fluid, fluid dynamics along with the structural mechanics?

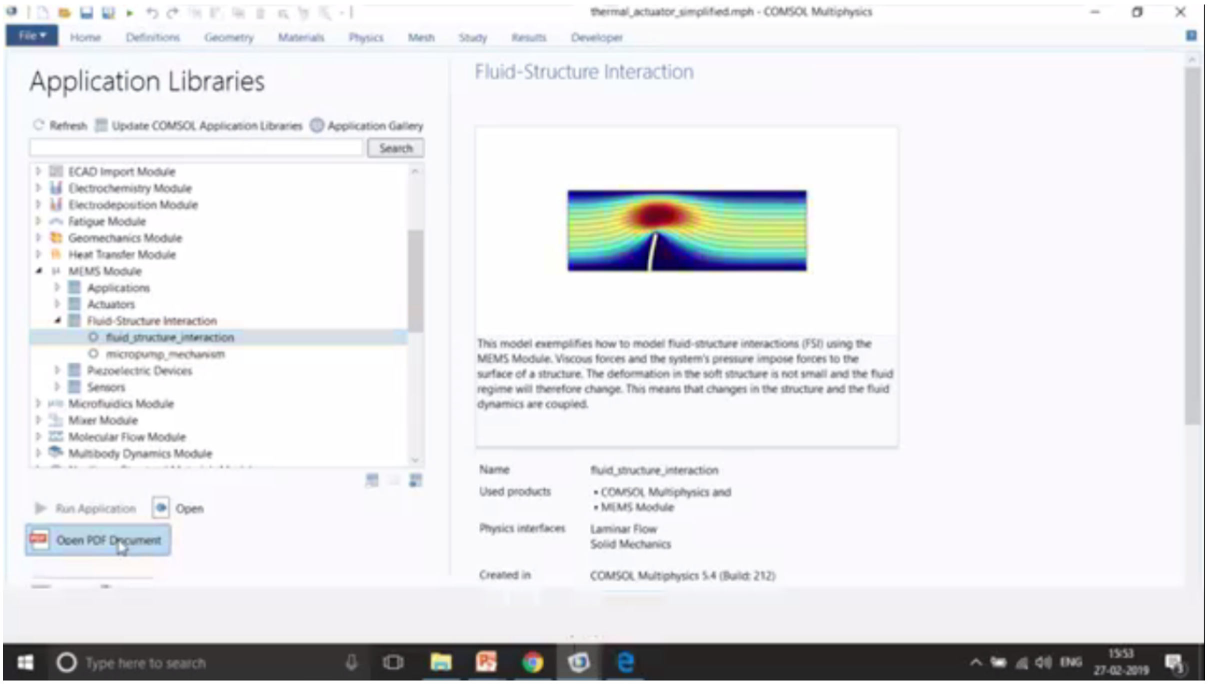
(Refer Slide Time 30:21)



So we have a couple of examples to do that. One of the example is a very simple flow of fluid through a channel from left to right, and there is an elastic material as you can see over here and it is going to bend. So, again, when we talk about the bend, we also need to take into account the change in the geometry with the help of change in the meshing. So you can see that the meshing is also going to change in this particular domain.

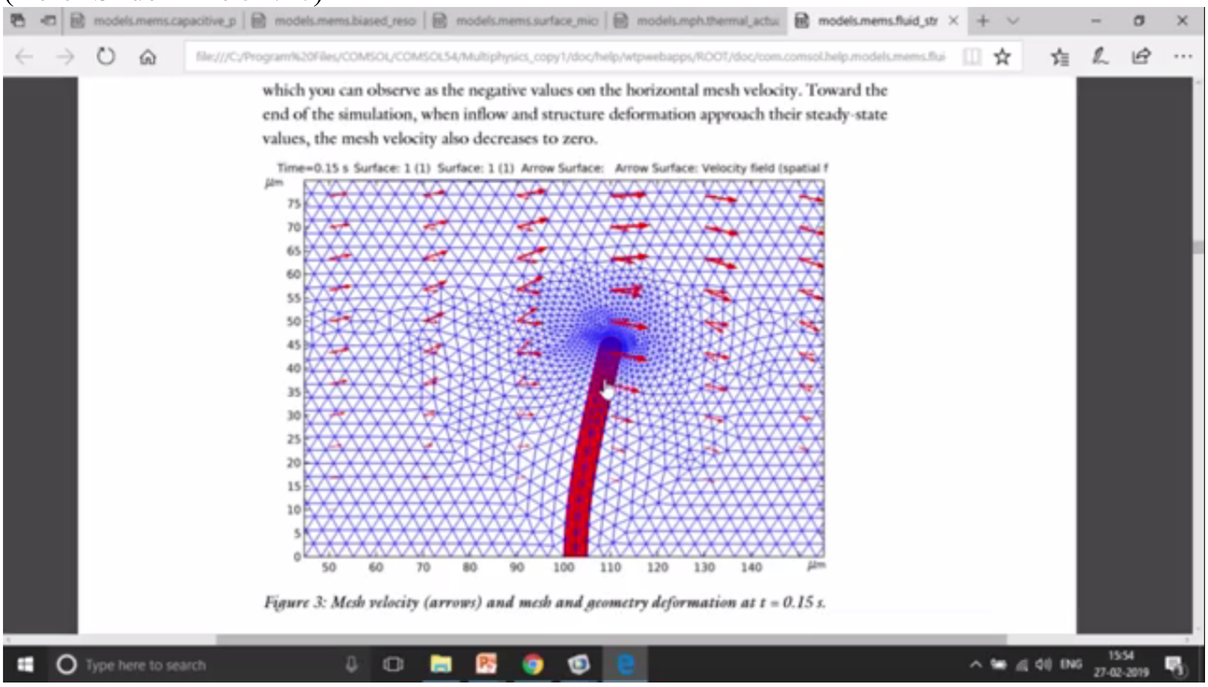
So let me just go and open this particular model. So let me just go over here and I go to the MEMS. I think it's available somewhere over here. Fluid-structure interaction, yes, it’s available over here. There are two examples. One is fluid-structure interaction. Another is micropump.

(Refer Slide Time 31:04)



In the Fluid-structure interaction, if you just open this documentation, you will see the Physics part and this is the most important part that I want to signify is the change in the mesh.

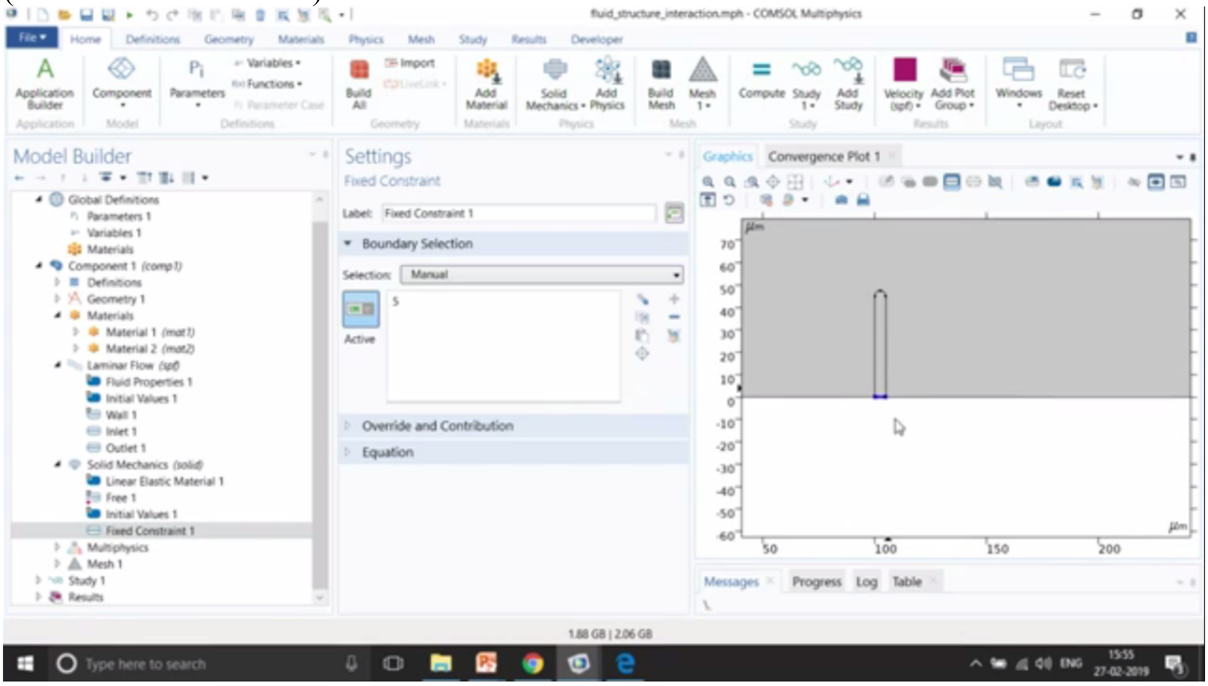
(Refer Slide Time 31:16)



So you can see this particular pillar is having a different mesh as compared to the original mesh which was actually vertical. So this is not any more vertical. It's actually tilted with a particular angle and that would only happen if you apply moving mesh or a deformed mesh.

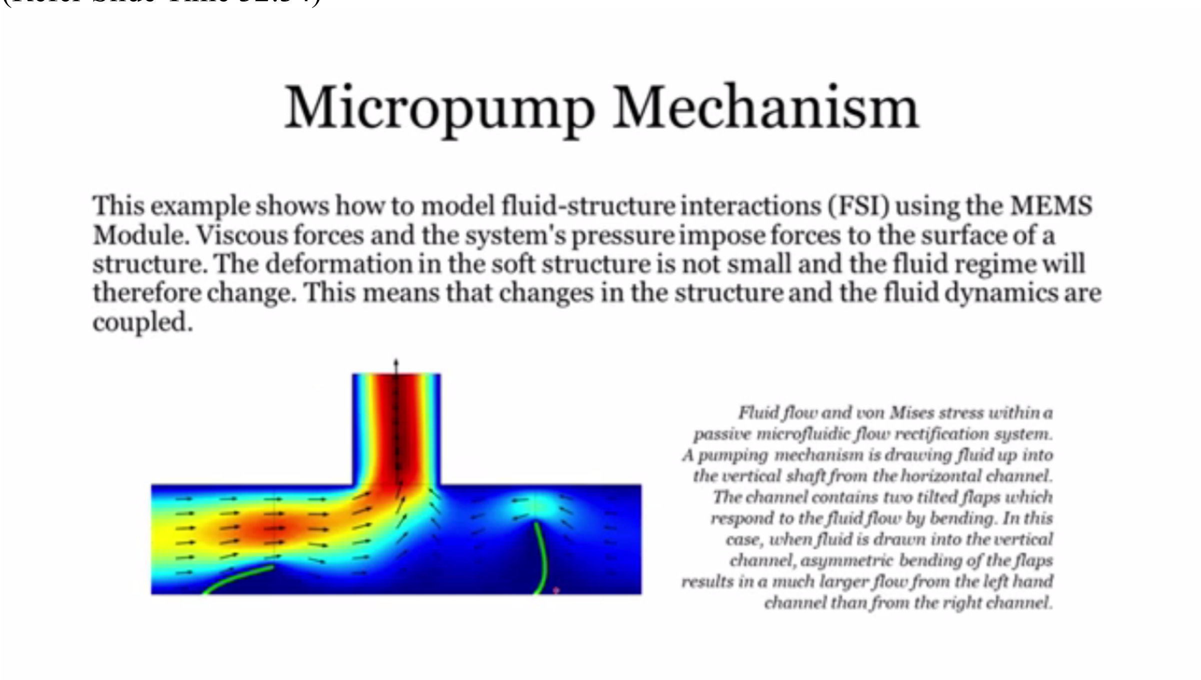
So just quickly go through the physics behind this. So this is the laminar flow. This is the particular structure. We have two materials. One is the may be water I guess and the second is the structural material over here. This is the pillar and we give an inlet fluid from the left side with the particular flow velocity and outlet on the right side. So the fluid is going to move from the left to the right and the forces are going to get exerted by this pillar.

(Refer Slide Time 32:11)



Such a mechanics is used to model the deformation of this pillar. A fixed constant has been given in the bottom part, so it cannot move. This particular boundary cannot move. The other part are free to move, right? The deformation is given using moving mesh. This is the most important part that, that needs to be taken into consideration. That is the move, the mesh is going to change, right?

(Refer Slide Time 32:34)

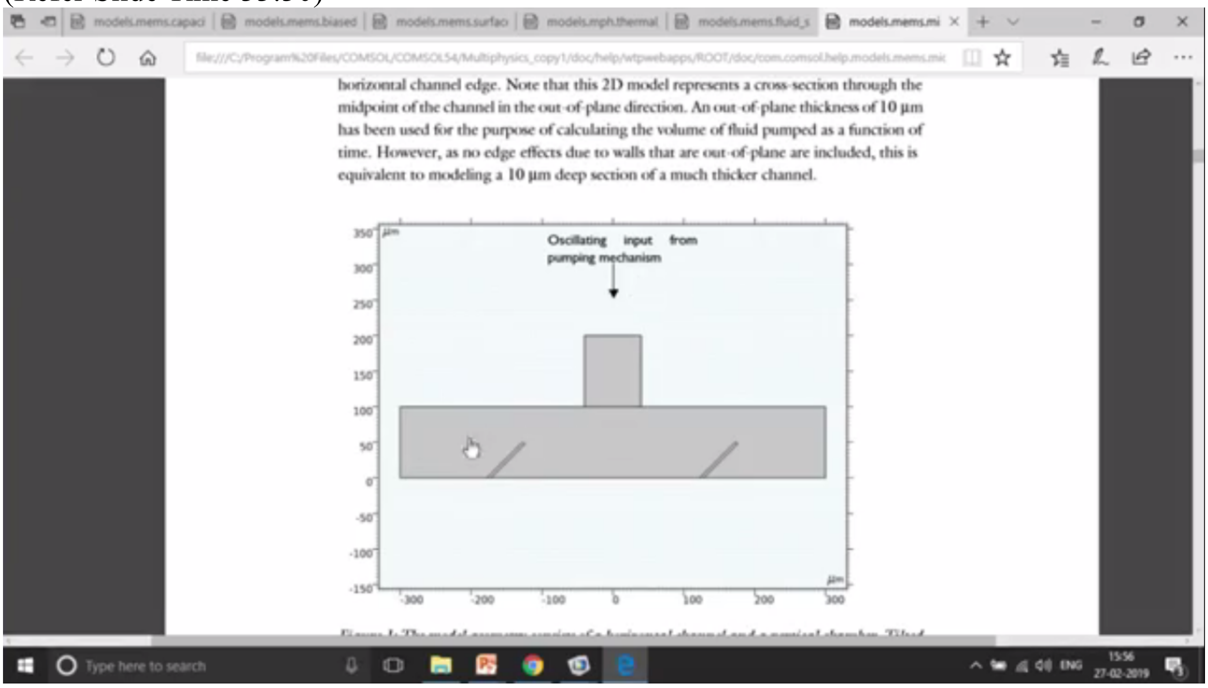


The next example is going to be the micropump mechanism example in which case we have an inlet on this left side and there is a pump on the top, and it’s going to, the main importance of micropump is that it allows the flow to in only a particular direction, right?

So over here there are two pillars that you can see which are actually bending, and based upon that it's going to store the fluid at the top and then it’s going to again refill. So the fluid is going to be filled on the top and then again it’s going to move towards the right side, but the back flow is kind of restrained.

So let me just open this particular model, how to this model, a very similar approach to what we saw before. So this is again the documentation. So I would suggest you to go through the documentation. This is the actual part.

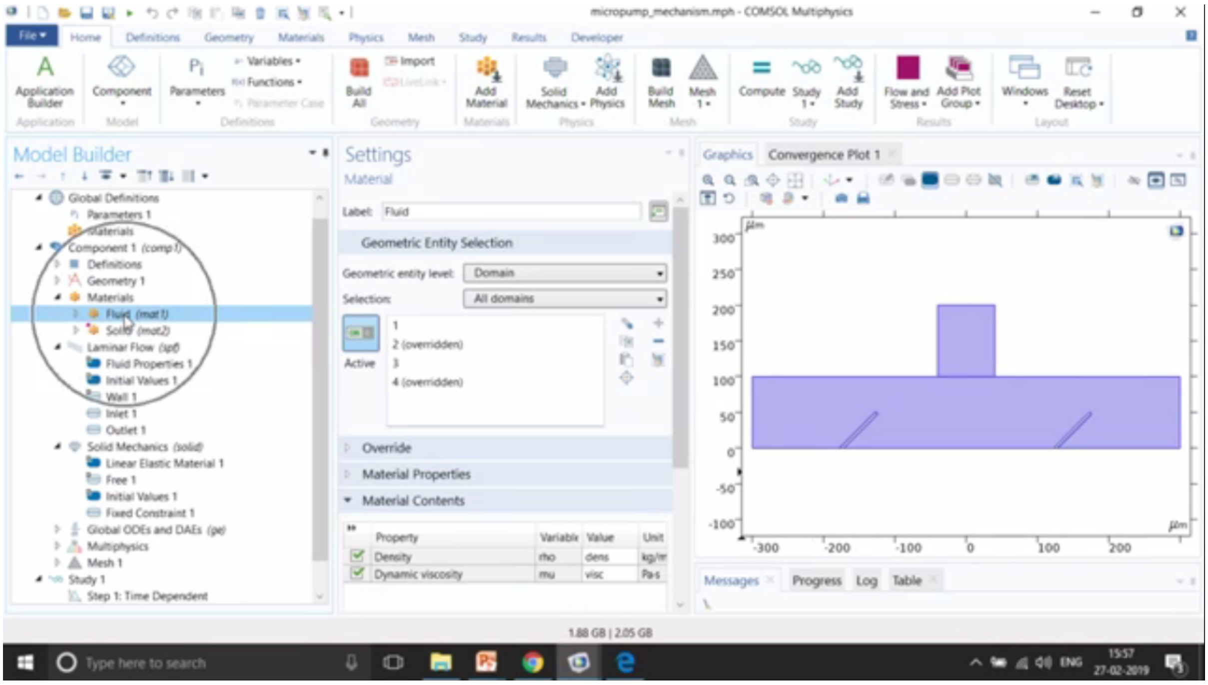
(Refer Slide Time 33:30)



So this is the actual input and we allow, you want to flow, have the flow in a particular direction, right? Again, the mesh is what is going to change at every instant.

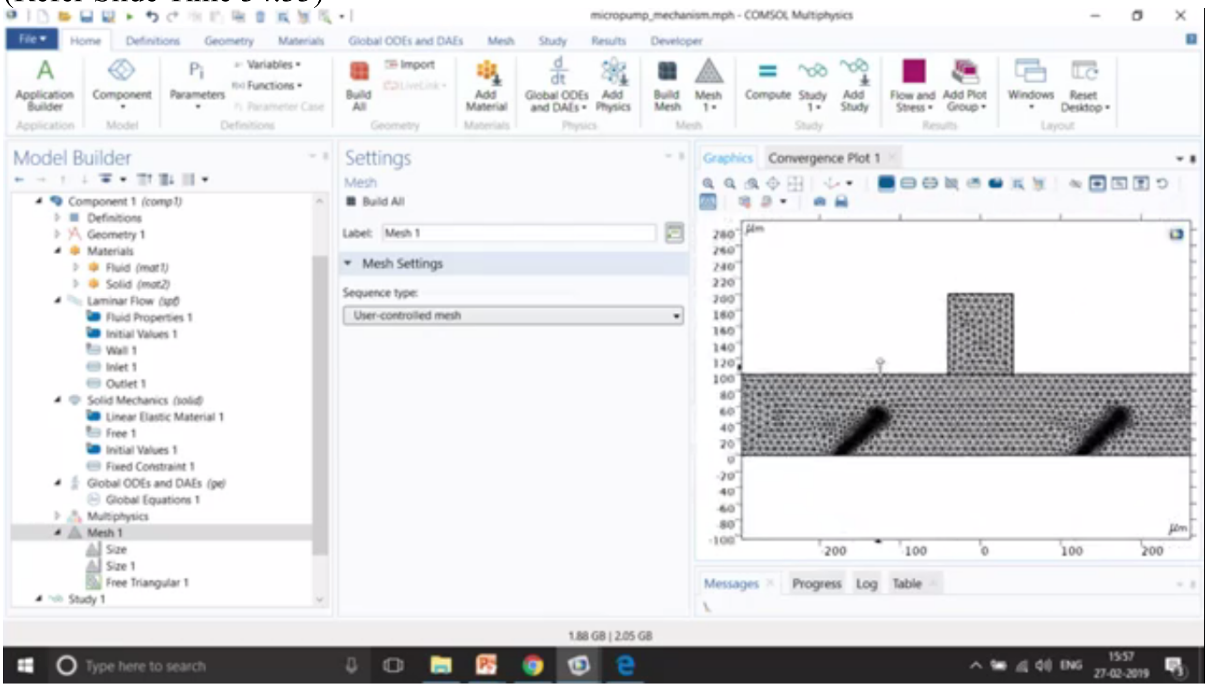
So let me just open this particular model. So you can see this is the time dependent simulation. So at each time step the mesh is going to be changed. Again, a laminar flow to understand the flow of fluid. Inlet is given over here. Outlet is given on the both the sides, and then the structural mechanics to actually have the fixed constant on the bottom part and the two pillars, which are having a particular angle is modelled using a particular value as you can see over here.

(Refer Slide Time 34:15)



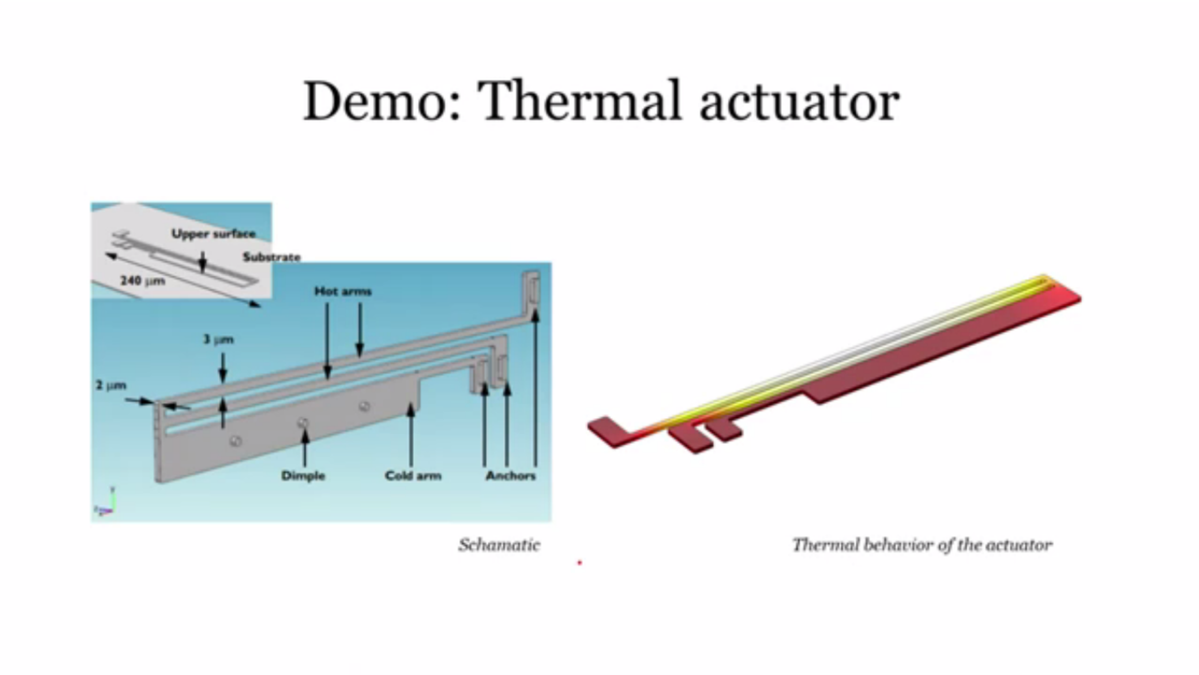
The rest of them is the fluid, which is defined by dense and visc, which is defined in the parameters. So you see the dense and the visk, that is the velocity and the density have been defined over here. This is to understand with what velocity is the fluid is moving and then finally the mesh that you can see over here.

(Refer Slide Time 34:33)



This is the actual mesh and then the deformed geometry using the moving mesh. Okay. And then you do a time dependence simulation to understand if it is moving with a particular, what is the flow rate that you can see from here. Okay.

(Refer Slide Time 34:53)



So next is the demo on thermal actuator. So I think that's how the time I have right now, right, till 4 o’clock. So I think I will stop.