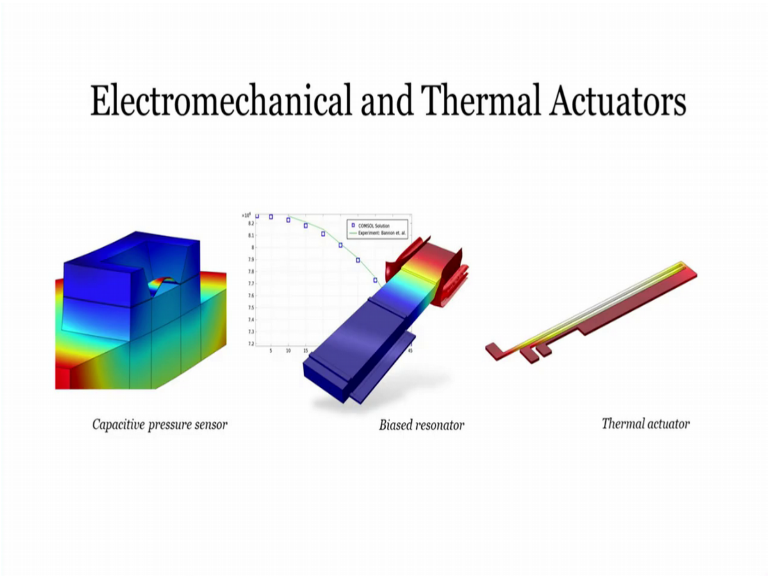
Electronic System for Cancer Diagnosis
Prof. Hardik J. Pandya
Department of Electronic System Engineering
Indian Institute of Science, Bangalore

Lecture - 68
COMSOL Examples for MEMS Applications (contd…)

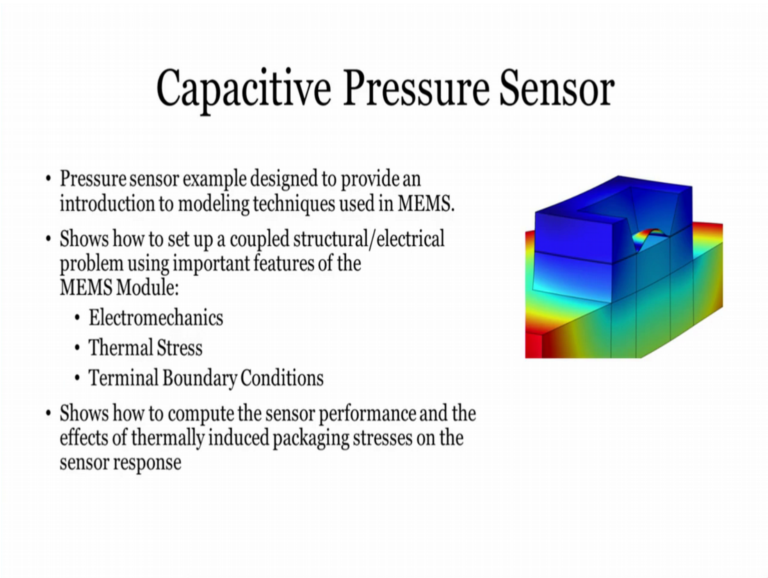
So, let me continue let us continue with electro mechanical and thermal actuator.

(Refer Slide Time: 00:33)



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 multi physics, how the coupling of structure mechanics with the electrostatics was occurring. In this example, that we are going to talk about, we going to take about how the activation actually happens and what are the physics behind that.

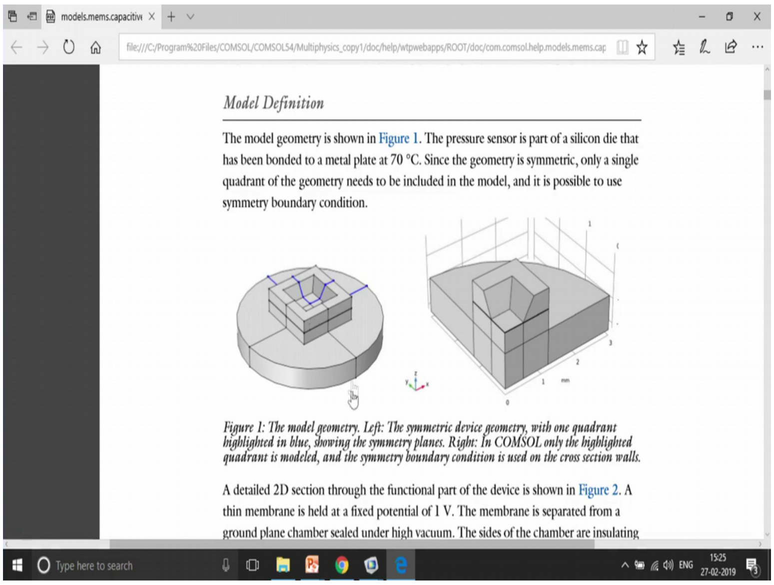
(Refer Slide Time: 00:57)



One of the example model that comes into picture is capacitive pressure sensors. These are regularly used in our mobile phones as you know. And, this is kind of touch senses that people want to model they want to have it on you there 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, again I go and search for piezo sorry not piezo, this is now electro mechanical systems. So, we have done away with the piezoelectric device.

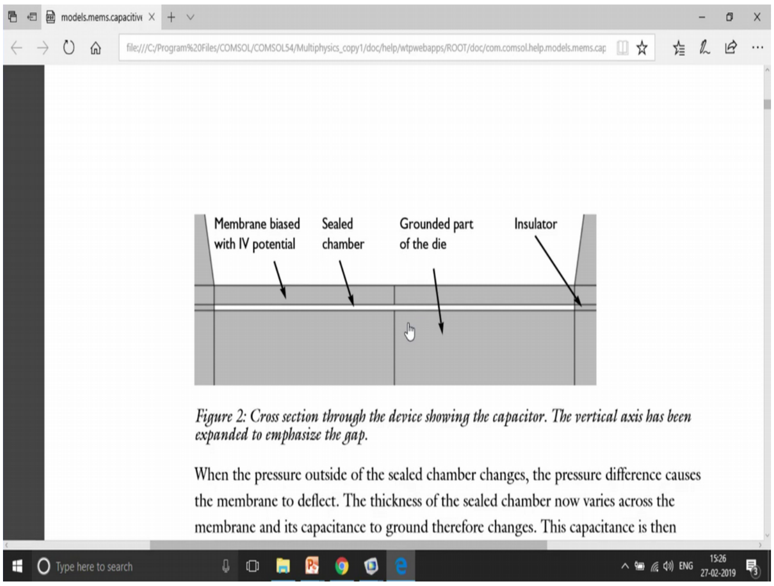
Now, we are going to move ahead with electromechanical actuators. So, let me go to actuators. And, I think it should be somewhere over here or it could be a part of structural mechanics module. So, if you are not able to search this search for a capacitive pressure sensor, so if you search searching for the first time, it may take some time, but then it will take very quickly. So, this is example which is all 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: 02:25)



So, again that you see over here, this is the actual geometry that you want to model ok. 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.

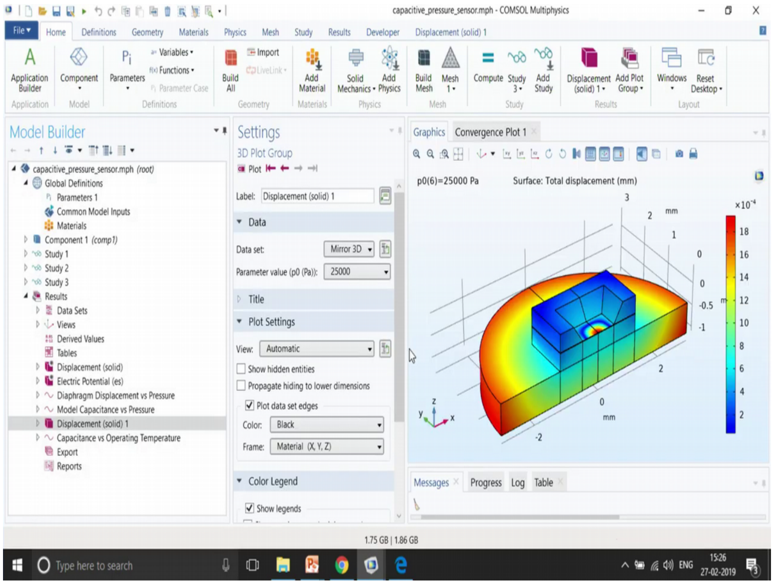
(Refer Slide Time: 02:58)



So, if you take a cross section over here you will see something like this right. So, you have the sealed chamber in the middle and then you have insulators on both the sides.

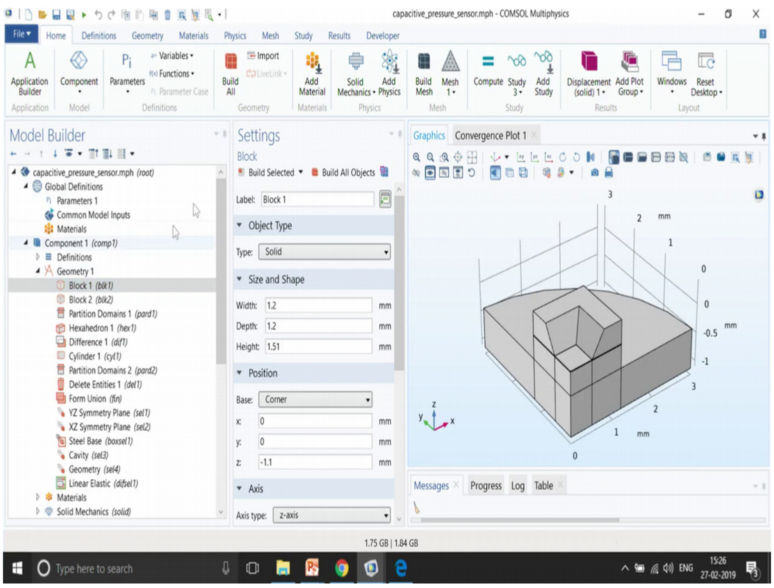
And then there are two membranes, one is the bottom part and the other is the top part ok. And this is actually works as a capacitor, because based upon the deformation that is going to happen over here. The change in capacitance is going to occur because capacitance is nothing but epsilon a by d. So, is a distance is decreasing. So, capacitance is going to increase right. So, let us go to the model.

(Refer Slide Time: 03:46)



So, let me open this model. So, over here again the first part is to make the geometry.

(Refer Slide Time: 03:49)



So, here you can see that how the geometry has been made. First making blocks and then partition domains and then finally, we have this particular domain right. So, we are going to model only a quarter part and then we are going to apply symmetry boundary condition on the right and on the left side ok.

Next part is the materials part very important part. Here we have used scalar material properties for the silicon as you can see over here and also the coefficient of thermal expansion comes in to the picture, so that means, this structure is also dealing with the thermal expansion. The next part is the vacuum part that happens between the two domains over here. This is the domain that is going to deform to the downward direction, and the capacitance is going to change. The next part is the steel domain that is in the bottom. This is the major part steel and this should be again we having a very high Young's modulus and very high density. Again everyone is a scaled over here.

The next part is electrostatic sorry the next part is the structural mechanics that is solid mechanics part. And over here we are giving a boundary load as you can see over here. So, boundary load has been given on this particular boundary ok. In this particular boundary, how much pressure has been given here it is p naught pressure that has been given, a p naught pressure has been defined over here 20 kilo Pascal.

So, give a particular load in the top and it is going to deform, because its an elastic material, it is going to deform. And along with the de deformation there is also thermal expansion that is because of the temperature there is going to be some a external temperature. So, here the temperature external is T naught, so it is around 20 degree, there 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 an while the bonding was taking place at that time some temperature would have been there. And because of their, they 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 issue also some kind of deformation that takes into picture because of thermal expansion.

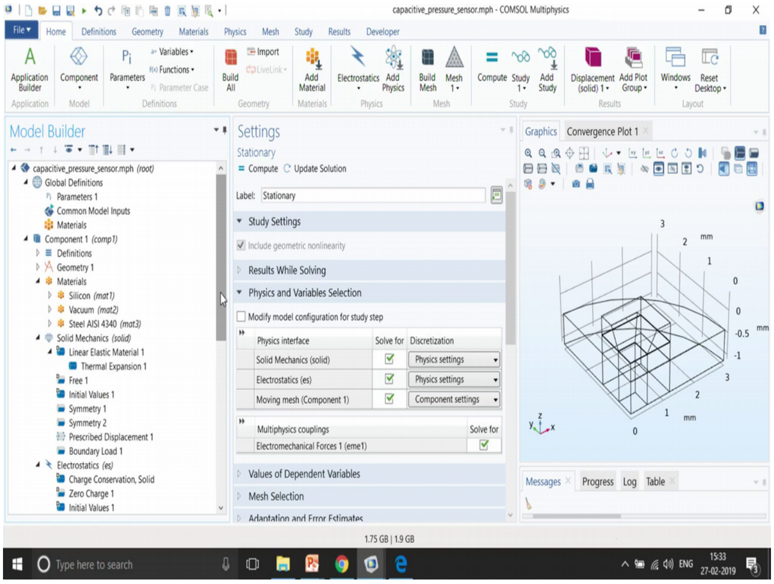
We definitely want to give symmetry boundary condition because we have modelling only quarter part. So, we have the symmetry a boundary condition on the right side and on the left side. And then we give prescribed displacement. So, why we are giving it is we are saying that it is not going to move in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

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

So, how do say signify what is a difference, here we will see the blue is completely filled cylinder, and over here the blue is filled only on the surface of the ground right. So, but in the middle it still vacuum right, so you can see over here right. So, in the bottom, you have for the ground and terminal is in the top. But in the in the middle, it still left. So, it means go to have a different views with that could help.

So, you can see this is a bottom part. And this is the top part terminal. Some part is all already there which is missing that is the vacuum. The next part is the multi physics. So, here we move from piezoelectric to electromechanical forces, which couples the solid mechanics with the electrostatics ok, so that is how it is done.

(Refer Slide Time: 08:21)



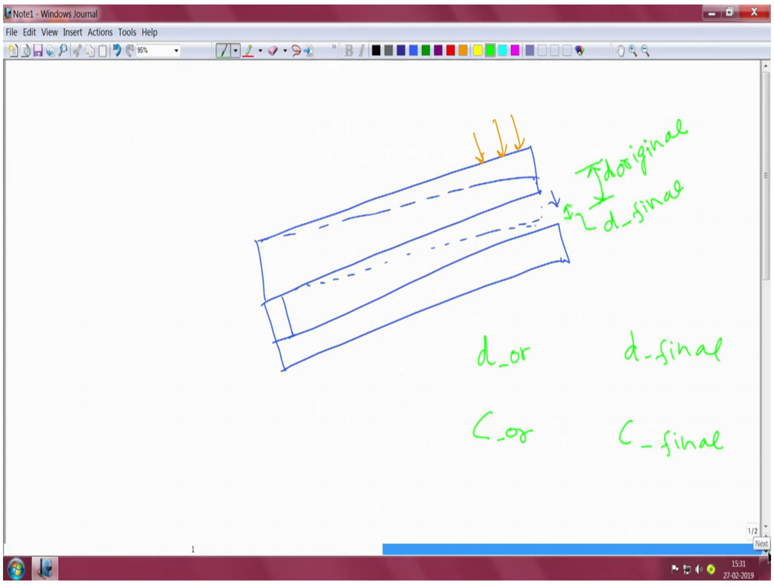
So, it has different types of studies. The first one is solves all the three of them as you can see over here. The solid mechanics electrostatics and the moving mesh. Moving mesh is the most important part now. Let me give you some introduction of moving mesh. So, for example, you have a particular geometry. 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 ok. And then we have one more bottom structure ok. 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 is going to bend like this right. So, it is going to bend to the bottom and we have two different values of d for example.

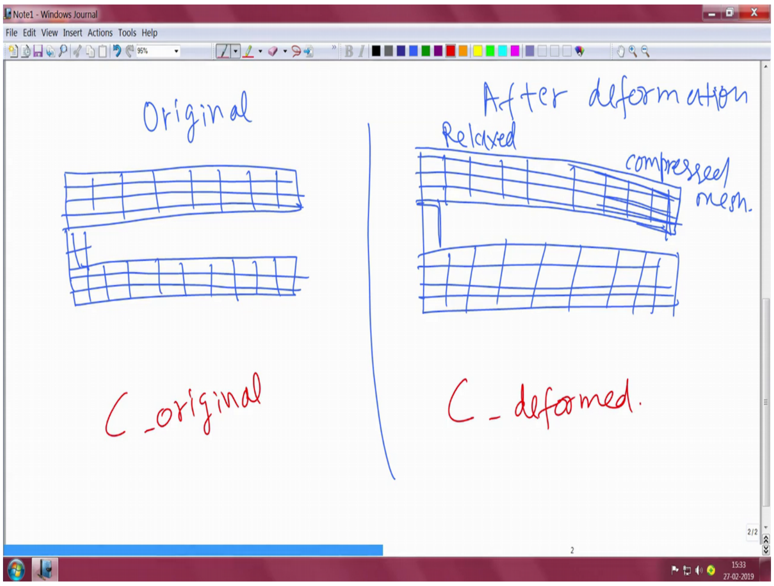
So, this was the actual d value that is the distance between the two plates. This is d original. And then we have after the displacement that let us define it as d final that the position at the final how much is the displacement right. So, we have d original and we have d final right. So, the d original will give you the original capacitance and the d final will give you the final capacitance after the deformation.

So, the most important thing that is happening over here 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: 10:44)



(Refer Slide Time: 10:53)



So, if you talk about two parts, so my first part is going to be the straight part. So, this is a just an explanatory diagram. This is not what is meshed over here. For an 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. After deformation once the load is given.

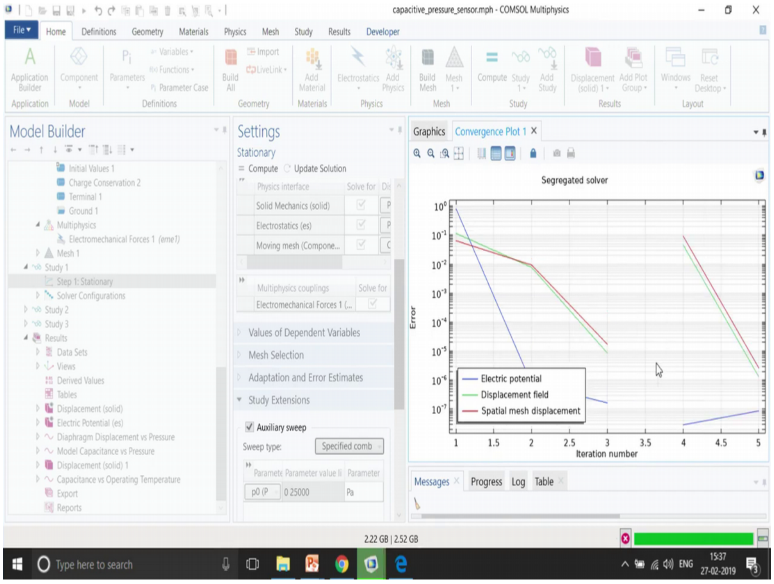
So, let me just after deformation. Your structure is going to actually deform. So, by deform I mean the bottom structure is going to remain the same, because we have given kind of a constraint at the bottom. But the top structure is going to move like this ok. So, you will see that your mesh also needs to get deformed 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 will be more relaxed; here it will be more relaxed mesh and over here it will be more compressed mesh. 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 you are discussing before that has to be taken into consideration using the moving mesh ok. 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 figure. And then after the deformation what is going to happen once you give the electrostatic signal.

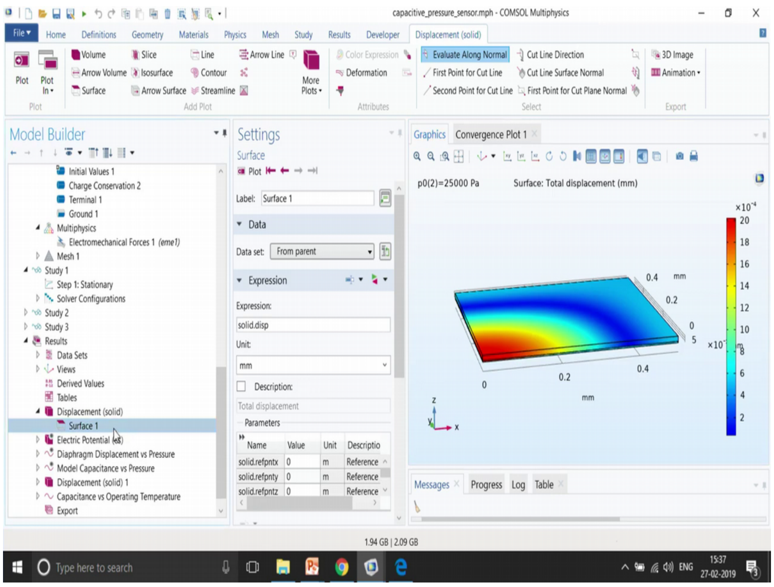
So, what is the capacitance of that? So, you have this is the ca the capacitance that the original geometry with that this will give and this capacitance that the deformed geometry will give is going to change. So, is very important that the mesh actually change, so that it will get reflected as a form of the capacitance ok. So, let us see how does that varies. So, over here they do a parametric sweep of different, different pressure; however, let us go with a very simplified way let us give 0 and 25000 just to see how the pressure varies.

(Refer Slide Time: 14:10)



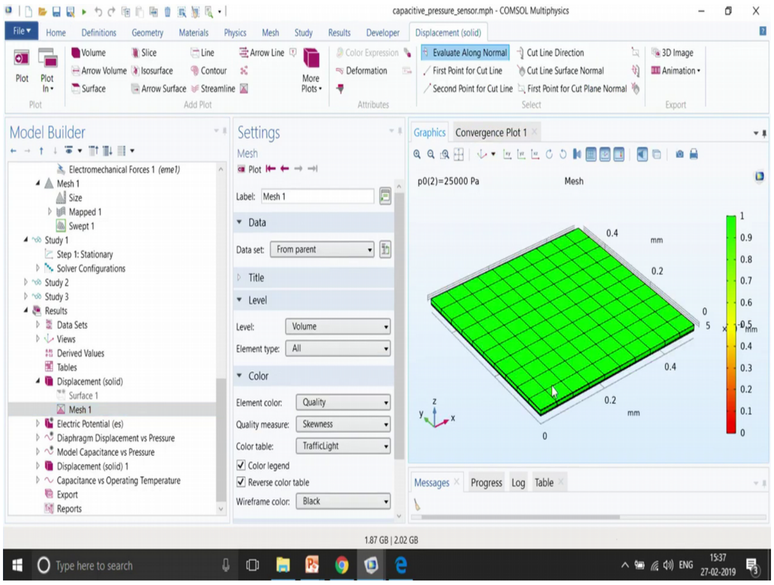
So, you can also see the convergence plot over here usually the errors a go below 10 power minus 3 to get it resolve ok.

(Refer Slide Time: 14:16)



So, we have the results. So, what I want to show case you are the mesh. So, I was just disable this; I will first plot the mesh. So, you can see that mesh is also there if you want to only mesh I mean you wanted to want to visualize the mesh. So, I just make a volume mesh and this is the actual mesh right that is coming from the mesh over here. 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 ok.

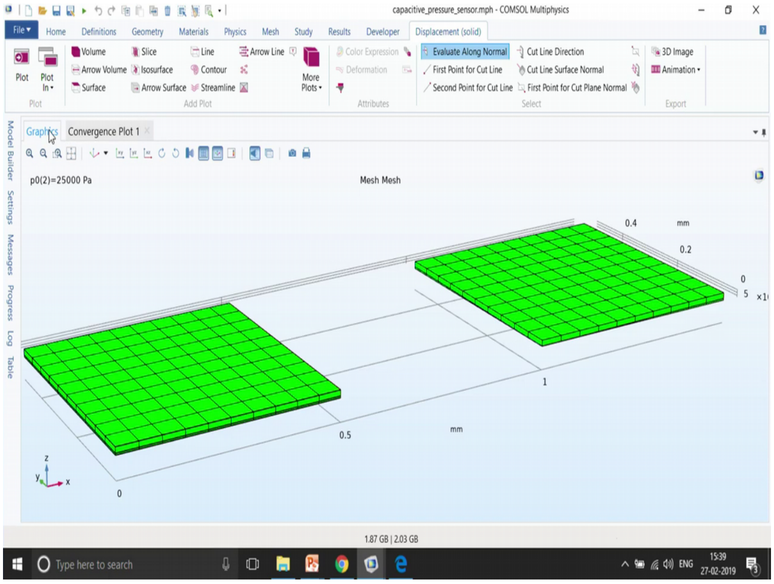
(Refer Slide Time: 15:04)



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 25000 kilo sorry Pascal's of a pressure. So, you can see the some deformation was there right. So, you can see the change in the mesh.

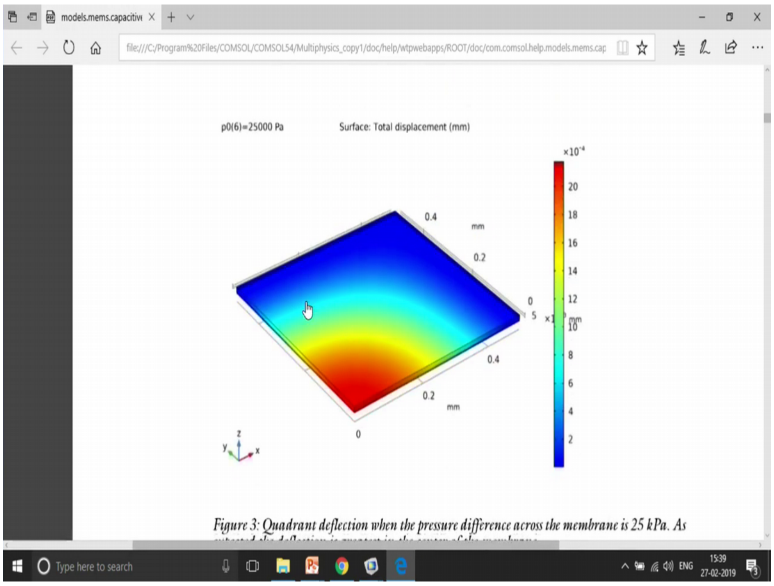
And this change in the mesh is then taken into consideration while we do the simulation. If you want to see its side by side that also is possible just duplicate it and have some kind of a deformation to shifted towards the right side that is in order of 1 mm. So, I just move it to the right side. So, I can visualize both of them together both the mesh.

(Refer Slide Time: 15:57)



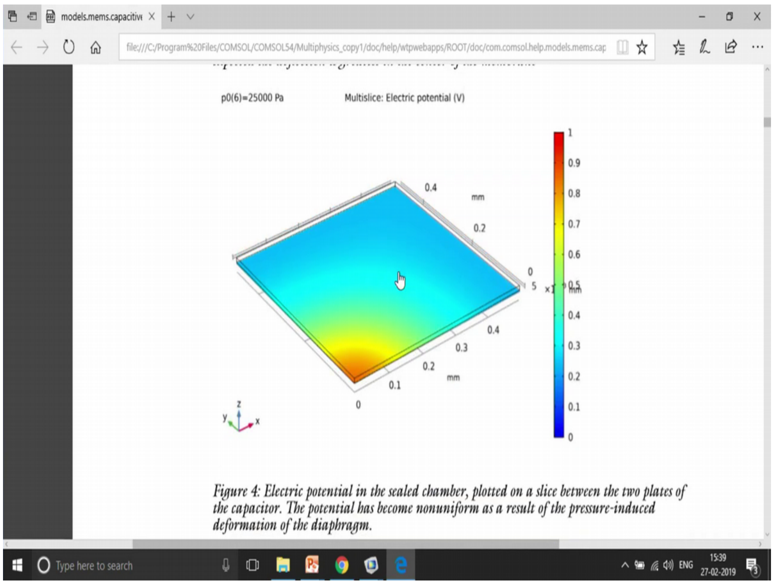
So, in the mesh 2, I just write as 25000 right. So, if I want to just maximize it, so you can see that on the left side is for a no deformation and then on the right side it is with the deformation. So, both are having a different mesh.

(Refer Slide Time: 16:19)

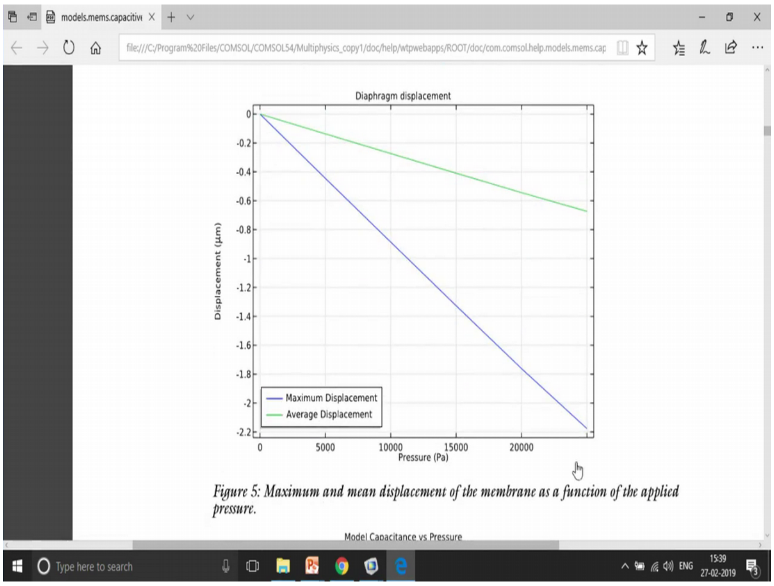


So you have the potential plot over here and then the deformation plot.

(Refer Slide Time: 16:22)

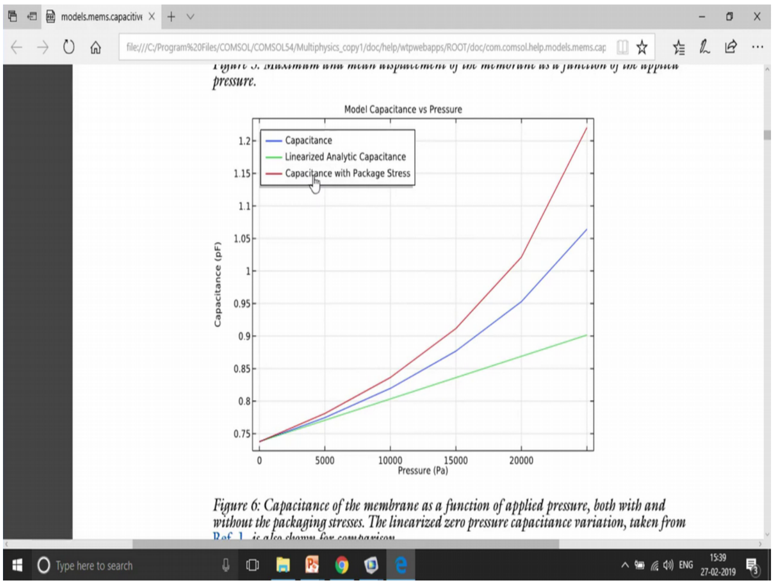


(Refer Slide Time: 16:25)



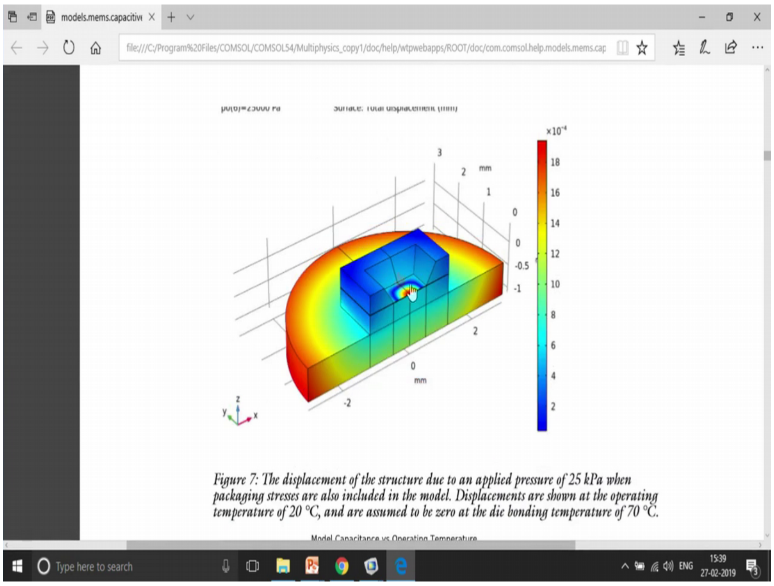
And, then you can also vary the pressure from 0 to 25000 and then see how much the change in displacement is going to occur at a particular point ok.

(Refer Slide Time: 16:38)



You can also compare the capacitance with the analytical capacitance, that is available as an equation and then see how much is it compare compared with this right.

(Refer Slide Time: 16:51)



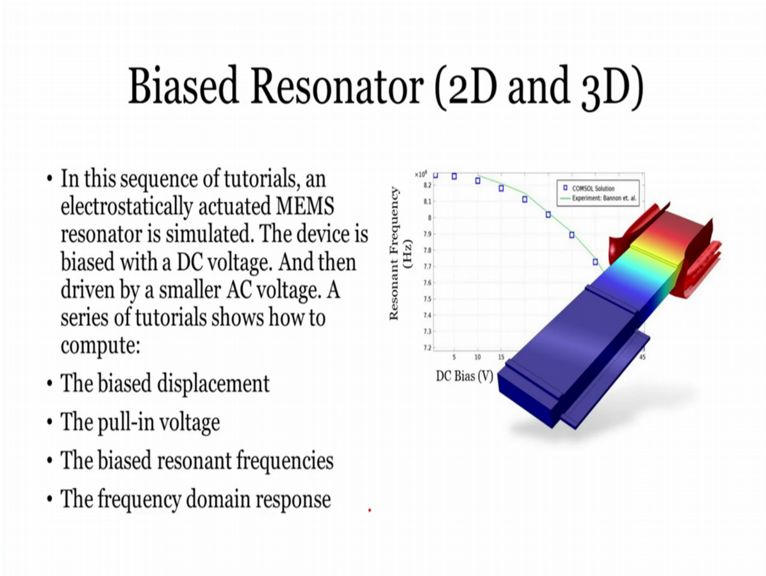
So, the complete simulation so, now I have done only for the quarter part. But if you want to simulate for the other part of the complete structure, you can also mirror the data set. To mirror the data set, you can just go to data set and use 3D mirrors. So, if this is my actual geometry, see this is my actual geometry that you can see over here, I can use the mirror to actually make it half of the geometry. Again use the mirror, so I can just use one more mirror.

So, let me go ahead and add one more mirror and use my study 2, study mirror this one. And then and to choose this zx-planes and I get the complete solution. So, I actually model only a quarter part, but I get the results 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 two data set that is nothing but this data set. Just use this one and just plot it and I get the result for the complete structure, where I have only model for a quarter of the structure ok. 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 a electro electrical physics and from there we understand what is the capacitance.

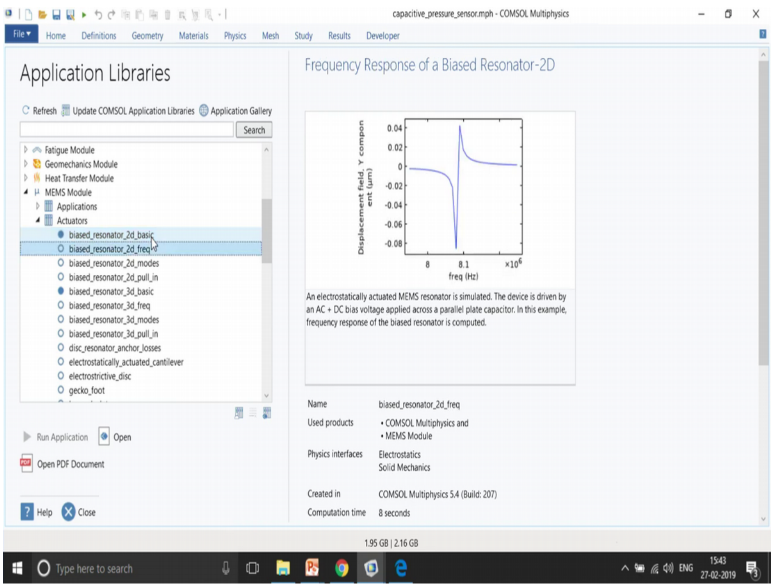
The way to evaluate capacitance is very simple. Just right click on the derived values, global evaluation, and just search for capacitance. You see that you can get the capacitance with the value of es C11. So, we solve for two pressures. So, as you can see over here, the capacitance for two pressure is changing. I can make it as a unit of pico farads, and then again evaluate. So, 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 is 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 up on the pressure as you can see over here.

(Refer Slide Time: 19:38)



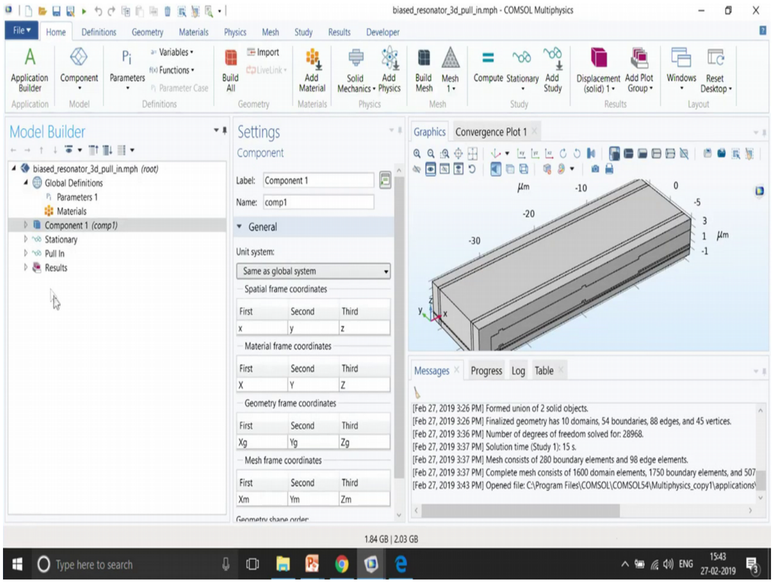
The next example is a bias resonator. This is an actuator beam kind of example where we give a particular pull-in voltage. And because of that pull-in voltage, how much de formation displacement or actuation of that cantilever beam is going to take place has been modelled. So, if you want to see that particular module, you can just go over here. I think it is available in MEMS module somewhere. So, let me just go through in the MEMS module in the actuators. So, you have many different kinds of analysis on bias resonators.

(Refer Slide Time: 20:14)



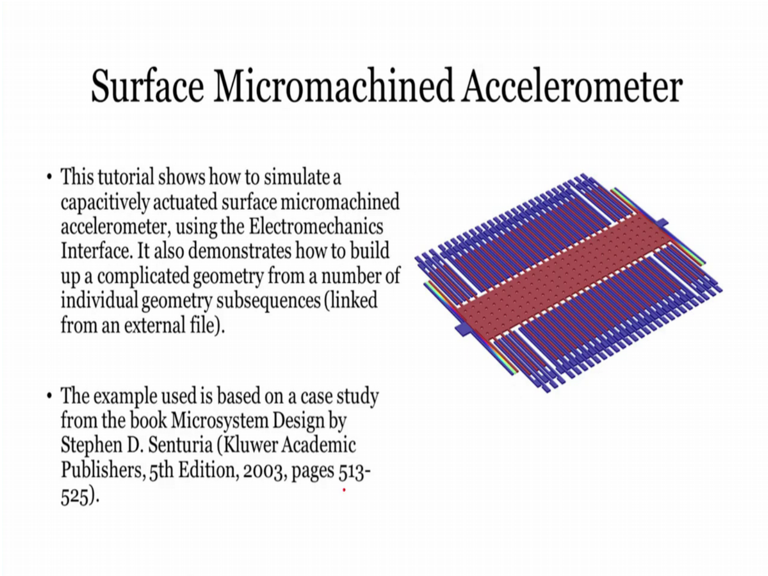
So, you can have particular frequency modules, where you want to know what kind of modes with which it will going to fluctuate, we have pull-in of this voltage I means at a particular voltage where it is going to touch the bottom plate kind of thing analysis, then we have 3D structures again right. So, you can actually open any of that module. First go through the documentation and then open the module ok. So, I will just open that particular module one of the module and explain it.

(Refer Slide Time: 20:50)



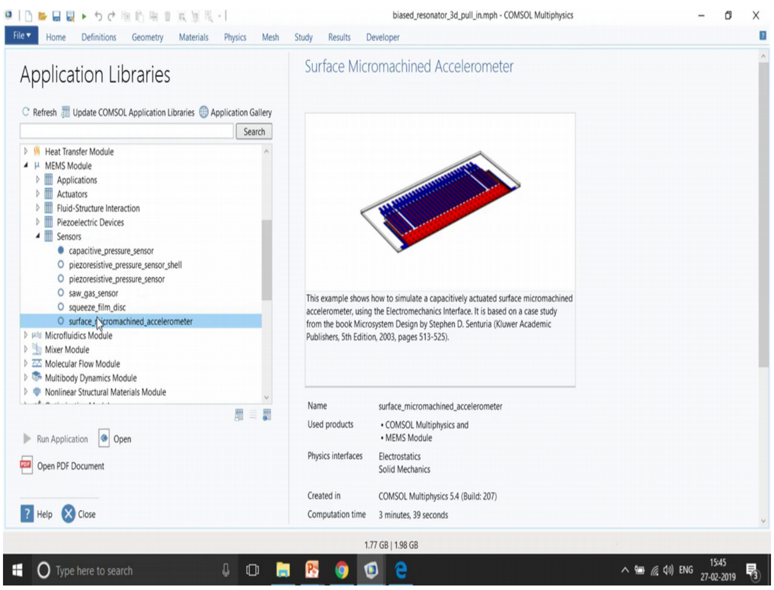
So, again geometry part is very important than the material part. So, here we have used a polycrystalline silicon over here. So, if I enable the wire frame rendering, it is poly crystalline silicon that you can see over here. And then we have silicon nitride in the bottom part, and silicon oxide over here and the rest of the domain is air domain ok. In structural mechanics or solid mechanics, we give a particular fix constant 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 a 2D resonators ok.

(Refer Slide Time: 21:53)



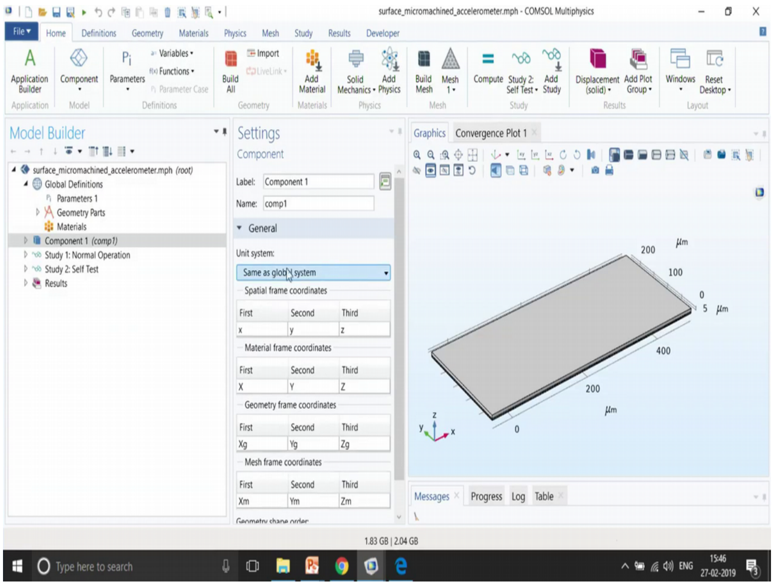
So, let us go ahead and this is an example model of an accelerometer. So, this is also available in the application library of COMSOL. So, if you want you can just go, so this kind of devices are usually used to understand the acceleration of a particular device may be your mobile phones you contain, so there are many games where which accelerometer devices have been used in cars, in vehicles, in aircraft, the accelerometers have been used a lot. And now they are also moving from accelerometer to sagnak interferometers, so which actually works on the basis of optics, but in this case it is purely electromechanical approach.

(Refer Slide Time: 22:46)



So, let me just go ahead and open this particular module. I have this module over here, somewhere. So, we have an example of surface micro machine accelerometer. The geometry of which is little bit complex. So, we have the proof mass, the folded spring, the building blocks that is electrode arrays, both the electrode arrays and then the analysis part. So, let me just open this module.

(Refer Slide Time: 23:10)



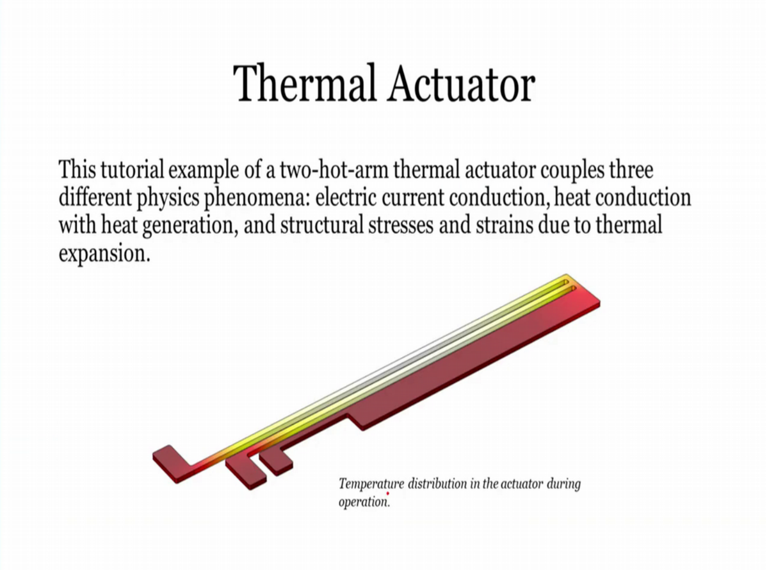
So, this is a structure that you can see over here geometry part has been modelled over here. So, it has been introduce from the geometric parts. So, there are many structure that are already available in COMSOL. So, if you want to understand what are those materials, so you can just right click on the 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, I 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 up on the focal point the complete design of lens is going to change.

If you talking about some kind of a beams and bolts, you can also look try to install from here. 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. So, over here they the module has introduced this kind of domains from the part libraries directly. And from there they have introduce that have called in this particular interface. And then applied the material properties that is air and polycrystalline in the bottom and then finally, the physics part that is the solid mechanics part that talks about a particular fix constants.

So, we give a fix constant at the bolts that you can see over here the blue part over here right. So, these are the different kinds of fix constant here, the mini you can see over here. So, all of them are bolts that is why fix constants have been given. The blue part signifies the fix constant. Then the symmetry part because we have module only half of the accelerometer, not the complete accelerometer that is why we have given 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. Then the electrostatics part then we have sense terminal and floating potential as we discussed before to withdraw a particular voltage ok. Then we have a multi physics. Coupling which couples the solid mechanic with the electro statics. And then we have two types of study both are stationary, 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: 25:55)



So this is then the last example is 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 two tutorial module also. So, let me just open this documentation. This is what demo that we will do in this section.

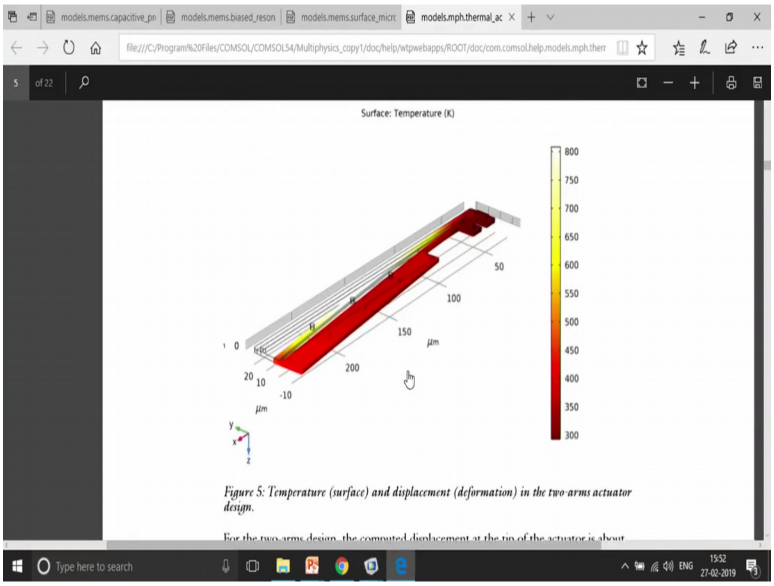
This is an thermal actuator that you can see over here. And these are the anchors, this is the cold arm, the dimples are there with which it is going to get attached with this is the main two hot arms where the temperature would be rising because of joule heating. And these are the boundary condition we give a particular voltage and ground it to the other side. We also give some temperature boundary condition that is fixed temperature on this boundaries and this boundaries and fix temperature at the dimples that you can see over here. And, then we give a roller boundary conditions that it can role while it is moving along laterally like up and down in this case.

So, just open this module. We have also going to do a demo on this module that would be after this session. And here you can see it is made of poly silicon right. And then we give so here there are three physics earlier we use to work with only two physics, now it is 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 the this particular boundary ok. And then ground at this particular boundary.

And then in the heat transfer we give temperature boundary condition that is the room temperature at these points and then we give heat flux. So, it will be also getting cooled with a particular convection. So, convict cooling. So, that is defined as h t s, s, h t c underscore s. This h t c underscore s has been defined somewhere over here as 0.04 watt per milli Kelvin over here right. So, this is heat transfer coefficient, so that is a upper part. And the bottom part is also getting cooled up with the different heat transfer coefficient that h t c underscore u s, where h t c underscore u s has been defined over here so that is given by divided by 2 ok.

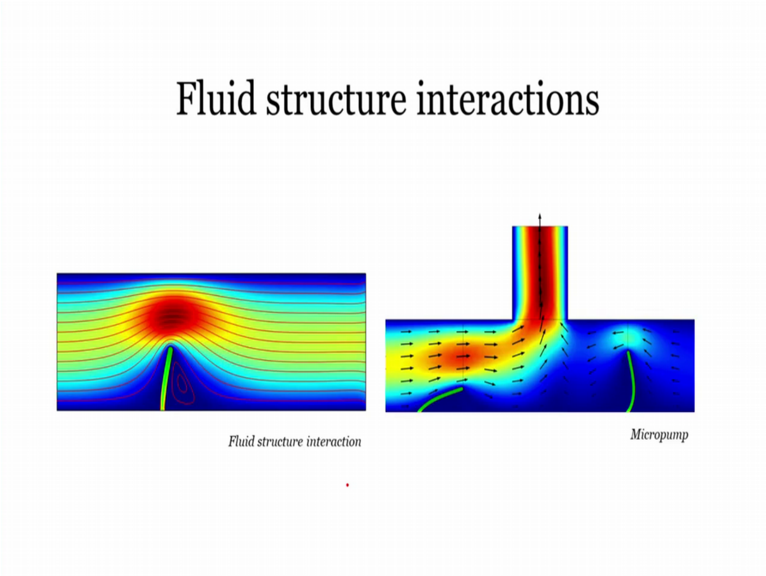
Let me just open it. So, it is 0.04. The upper part 0.04 divided by 2 microvolt micrometres and the bottom part is 0.04 divided by 100 micrometres. So, the top part is more as is getting cool quickly as compared to the bottom part. The last part is the solid mechanics part of course, we going to give some kind of fixed constant. It is given somewhere over here as you can see over here in the bottom. This is a three places where fix constraints have been given. And, then the electromagnetic heating is taking into consideration that is actually couples the electric currents with the heat transfer in solids ok. And you can just go to study and get the results.

(Refer Slide Time: 29:27)



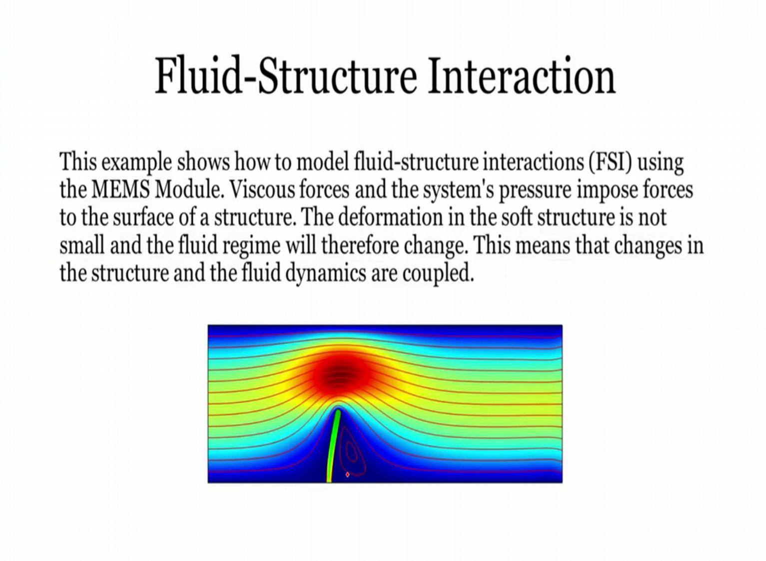
The results actually looks like this. So, you will see that the deformation is actually going to takes place like this. We have given a roller boundary condition that is why it cannot move up and down, but it is allowed to move in y direction right. It is not allowed to move in z direction, but allowed 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 a the red structure that you can see ok.

(Refer Slide Time: 30:05)



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

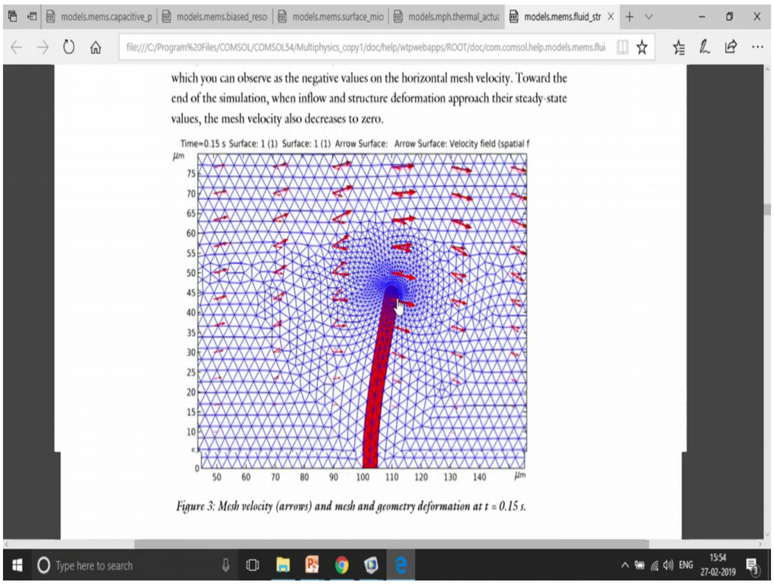
(Refer Slide Time: 30:21)



So, there are couple of examples to do that. One of the example is a very simple a 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; change in the geometry with the little bit 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 module. So, let me just go over here and I go to the MEMS I think it is available somewhere over here. Fluid structure interaction is available over here. There are two examples; one is fluid structure interaction, another is micro pump. In the fluid structure interaction, if you just open this documentation, you will see the physics part.

(Refer Slide Time: 31:11)

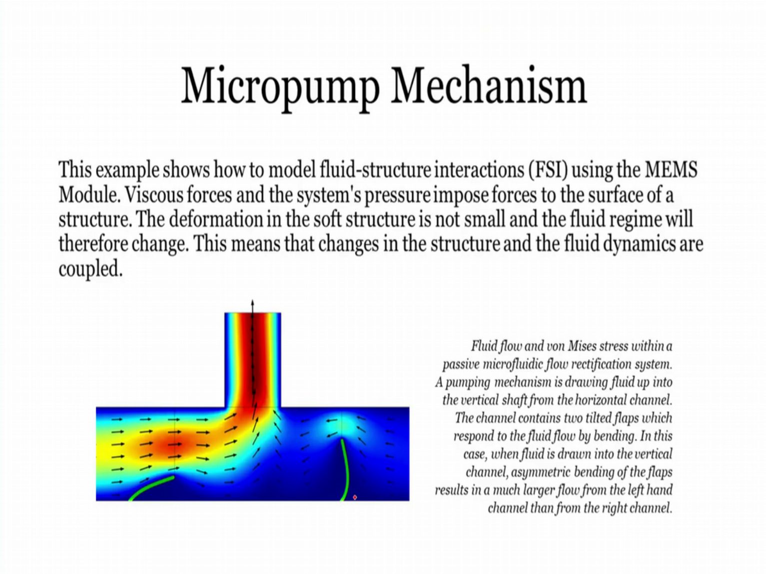


And this is the most important part that I want to signify is the change in the mesh. So, you can see this particular pillar is having different mesh as compared to the original mesh which was actually vertical. So, this is not anymore vertical, it is actually tilted with a particular angle and that will only happen if you apply moving mesh or a deform mesh.

So, just quickly go through the physics behind this. So, this is a laminar flow. This is a particular structure. We have two materials one is the may be water I guess and the second is the structure material over here, this is a pillar. And we give an inlet fluid from the left side with a particular flow velocity and outlet on the right side.

So, the fluid you are going to move from the left to the right. And the forces are going to get exerted by this pillar; so solid mechanics is used to module the definition of this pillar. A fix constant has been given in the bottom part. So, it cannot move this particular boundary cannot move. We other part are free to move. Why if the deformation is given using moving mesh, this is the most important part that needs to be taken into consideration that is the move the mesh going to change right.

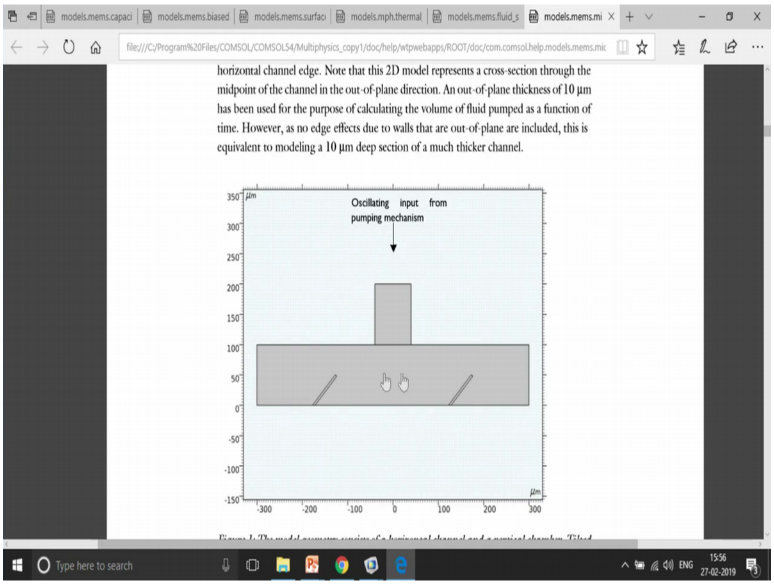
(Refer Slide Time: 32:35)



The next example is going to be the micro pump mechanism example in which case we have an inlet on this left side and there is a pump on the top. And it is going to the main importance of micro pump 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 is going to store the fluid and at the top and then it going to again refill, so it is the fluid is going to refill on the top and then again it is going to move toward the right side, but the backflow is kind of a restrained.

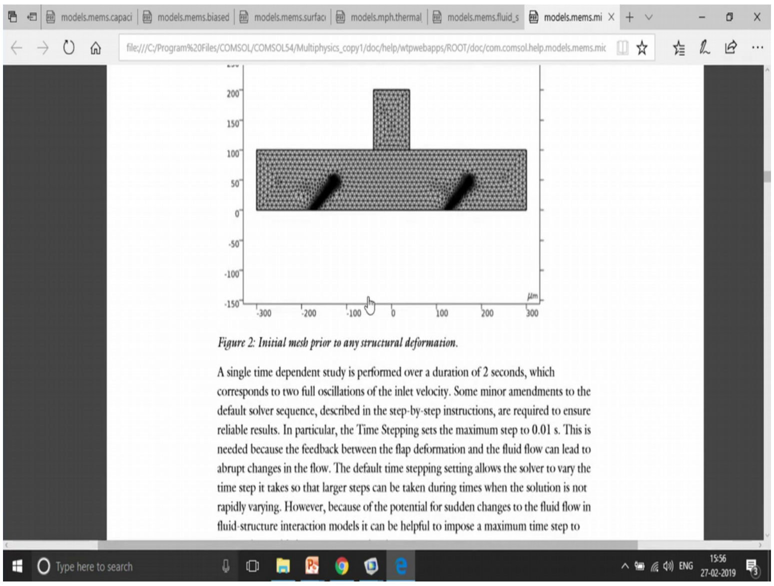
So, let me just open this particular module how it is modelled, a very similar approach to what we saw before. So, this is again the documentation services you to go through the documentation.

(Refer Slide Time: 33:28)



This is the actual part. So, this is the actual input and we allow you want to flow you have the flow in a particular direction right.

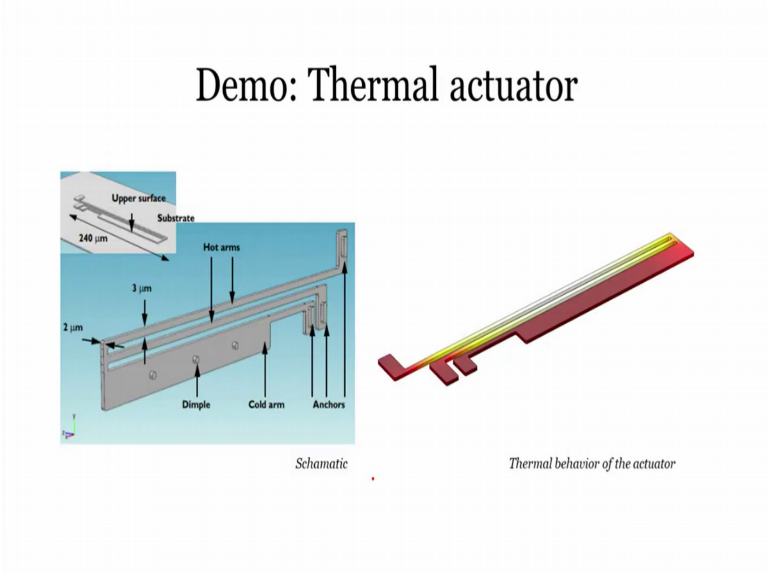
(Refer Slide Time: 33:37)



Again the mesh is what is going to change at every instant. So, let me just open this particular module. So, you can say this is a 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 at the fixed constant at the bottom part and the two pillars which are having a particular angle is modelled using a particular values as you can see over here. The rest of them is the fluid is defined by dense and visc which is defined in the parameters.

So, you can see the dense and the visc that is 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 this is actually mesh and then the deform geometry using a moving mesh ok. And then you do a time dependent simulation to understand if it is moving with a particular a what is the if the flow rate that you can see from here.

(Refer Slide Time: 34:52)



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

So, I think I will stop.