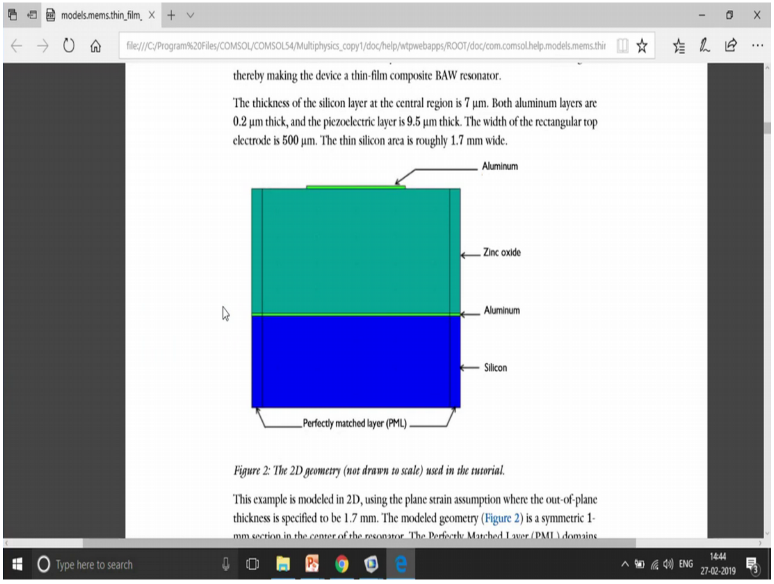
Electronic Systems for Cancer Diagnosis
Dr. Hardik J. Pandya
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

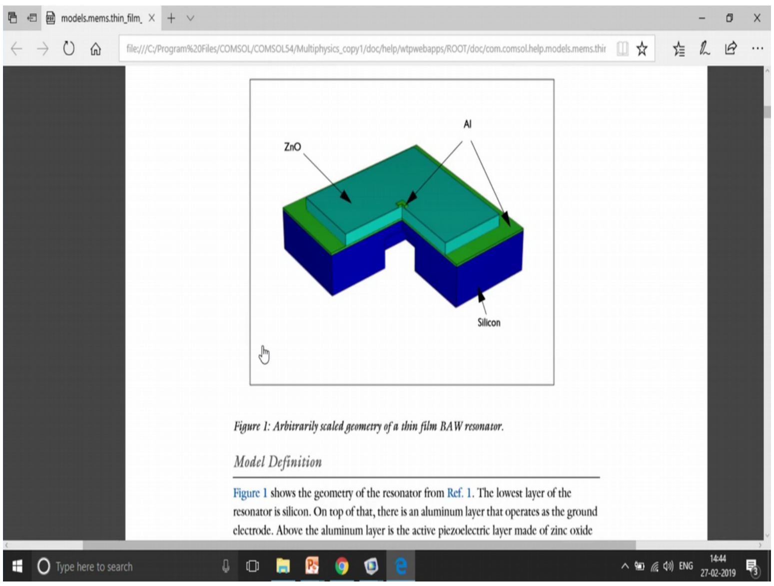
Lecture – 67
COMSOL Examples for MEMS Applications

(Refer Slide Time: 00:27)



And, in addition to it as we saw in the earlier example periodic structure were modeled right.

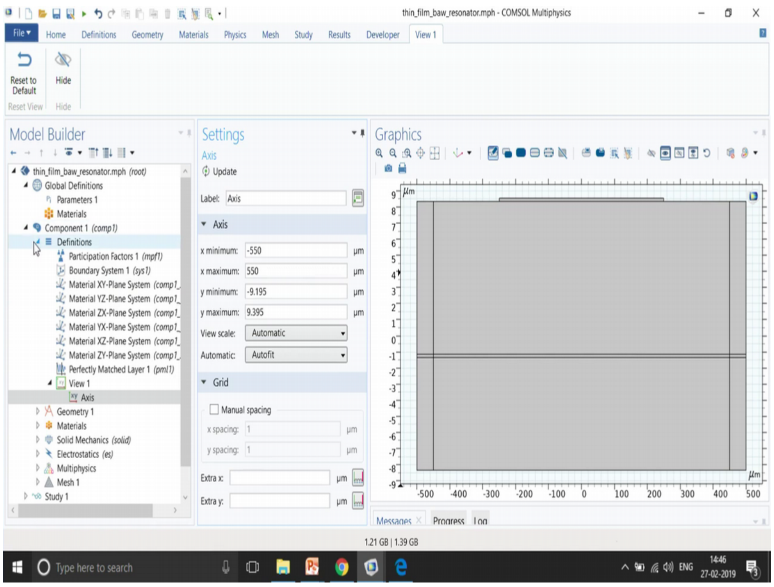
(Refer Slide Time: 00:37)



But, in this case there is no case of periodicity. Is not the above structure of aluminum, is not actually getting repeated. In this case hence we have use PMLs on the left and right. So, the force is that are going to go for the electric currents that are going to flow are not going to get reflected back from these boundaries. We should not forget that this whatever we are doing is a mathematical approach of actually solving the physical problem.

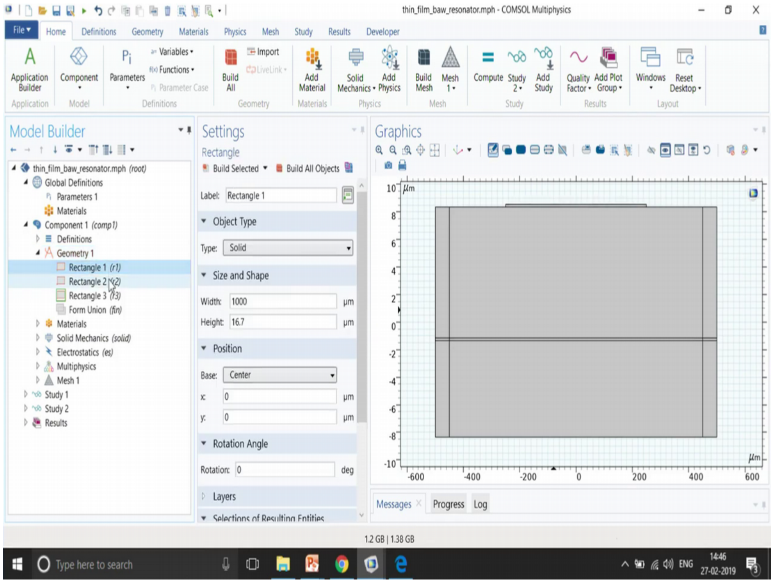
So, we need to terminate the boundaries very effectively one way to terminate those boundaries is using a PML domain and as we will see that there is a very particular type of machine which is required for PML that is a map mesh or sweep mesh that will talk about in sometime. So, this is the actual structure aluminum zinc oxide, aluminum silicon. The same thing in 2D has been modeled as aluminum zinc oxide aluminum silicon and finally, PML on the left and the right side.

(Refer Slide Time: 01:46)



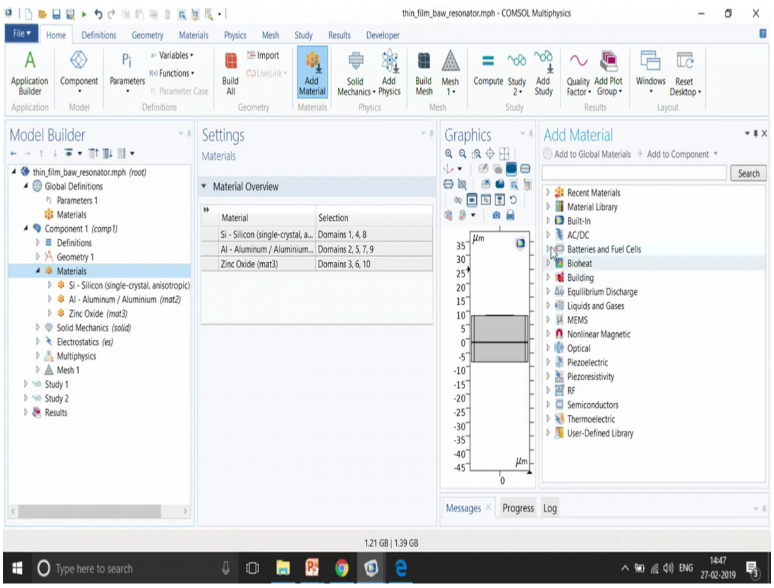
So, now let me go and open the model. So, right now you can see that your material this is actual geometry. So, in COMSOL you can also improve the view of your geometry by going to this definitions, view the view in go and axis and view scale as automatic. This will actually scale your geometry to little bit large. So, your actual geometry size remains the same, but the view of your geometry actually improve. So, that you can select those particular domains try to understand what physics you want apply.

(Refer Slide Time: 02:29)



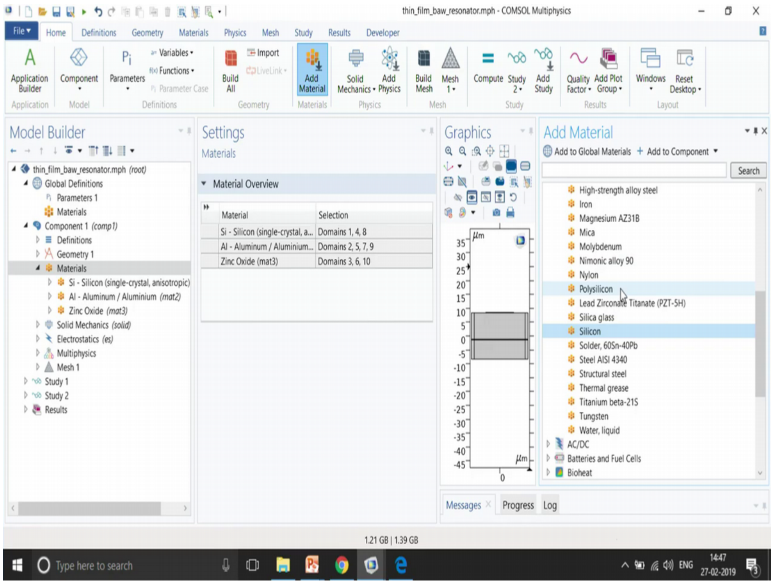
So, first thing is to make the geometry. So, I use rectangular 1, rectangle 2, rectangle 3 to make the geometry. The next part is the materials. The most important part is the materials.

(Refer Slide Time: 02:34)



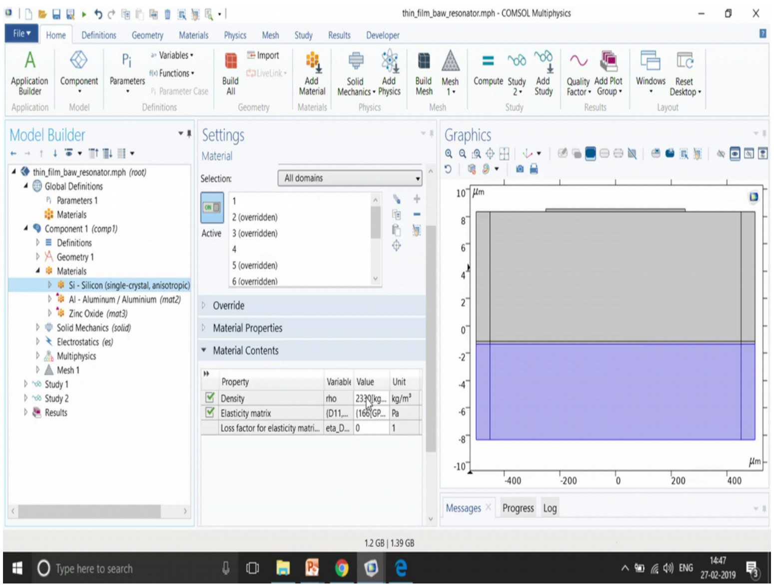
Here the silicon material has been modeled. So, we already have many different materials. So, if we just right click on materials, add material from library and you will see that there are many different materials. And, most of the times you would get the materials that you require from this material library. So, you have in piezoelectric. There are many different types of piezoelectric material available as you can see over here.

(Refer Slide Time: 03:08)



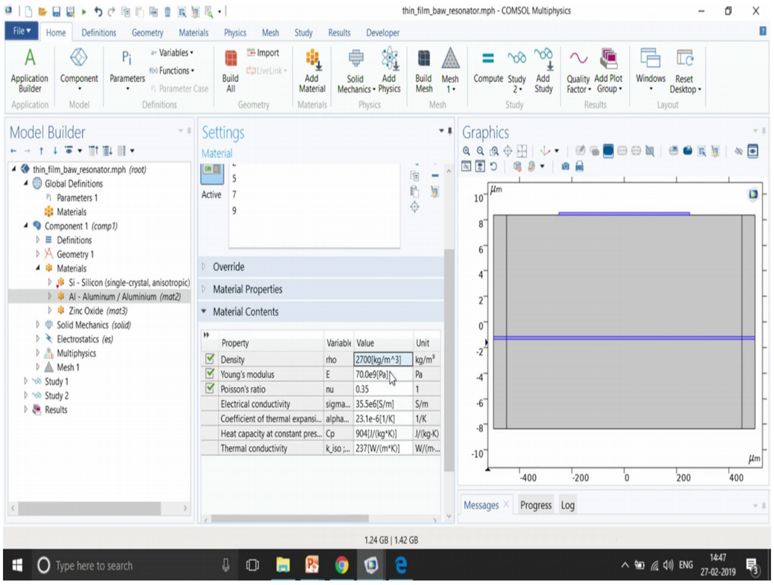
Silicon is available in the built-in. So, we will have a silicon over here somewhere your silicon over here. In addition to it we also going to do demo on thermal actuator and polysilicon is going to be used in thermal actuator.

(Refer Slide Time: 03:25)



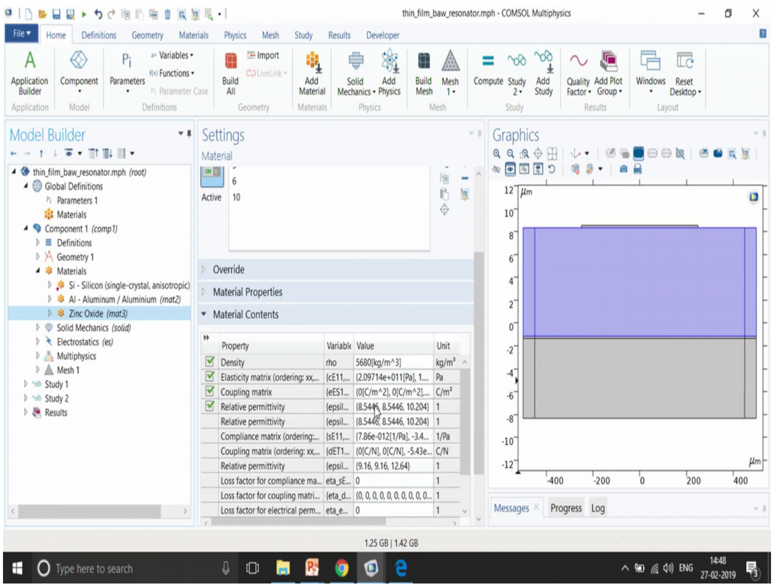
So, let us go to silicon that is the bottom part; and you can see the density and elasticity matrix which with which the silicon has been modeled again, a tensor with which the elasticity mass matrix has been defined for silicon.

(Refer Slide Time: 03:41)



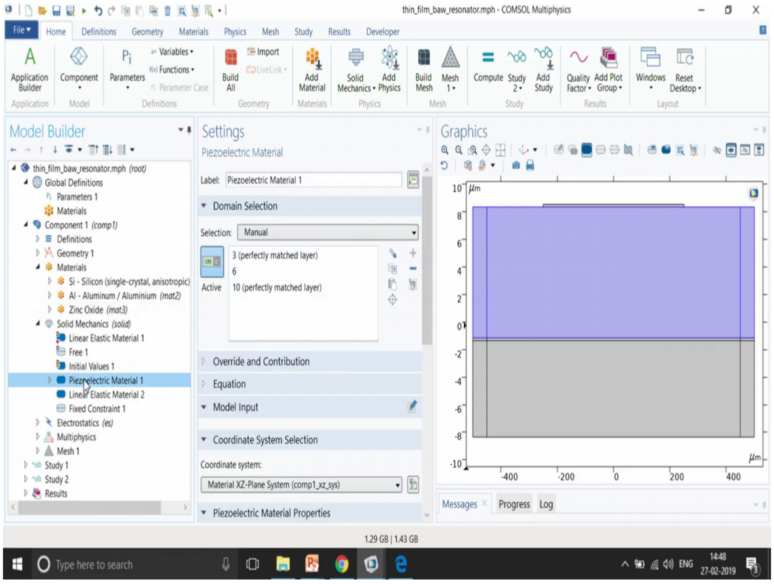
In the thin aluminum metal contacts, this we would see over here. This is again a scalar material properties and, then the zinc oxide material that you can see over here.

(Refer Slide Time: 03:56)



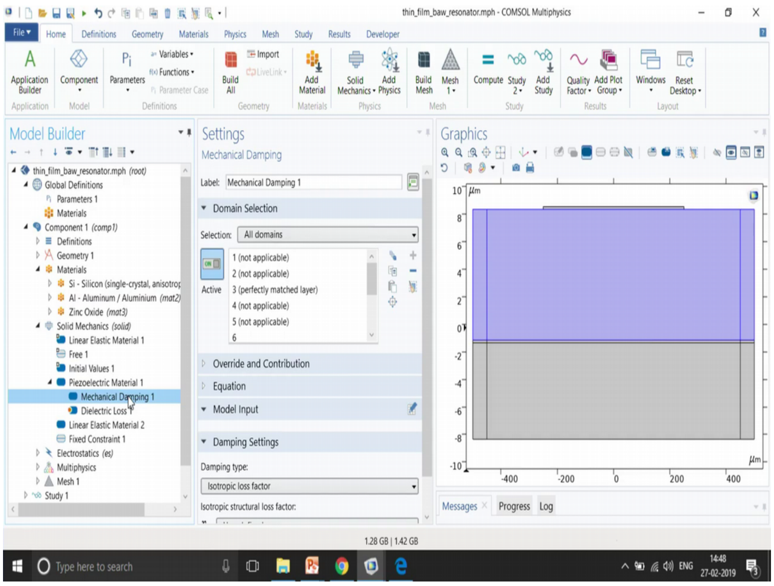
This is again elasticity matrix coupling matrix is a tensor and again the relative permittivity is n anisotropic which is more in z-direction as compared to x and y.

(Refer Slide Time: 04:14)



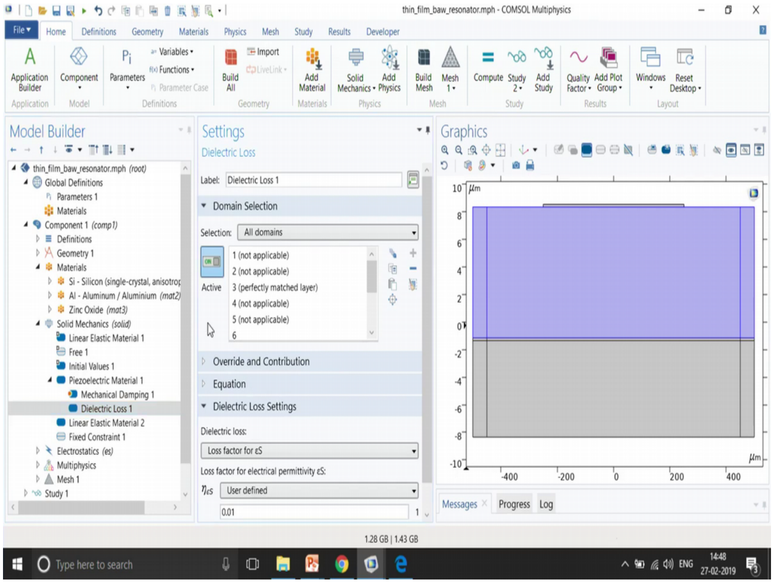
The physics setup is should be simple. So, we are using for zinc oxide you can see over here. We are using a piezoelectric material property over here and then you are using stress charge form and, in addition to it there are some kind of losses.

(Refer Slide Time: 14:28)



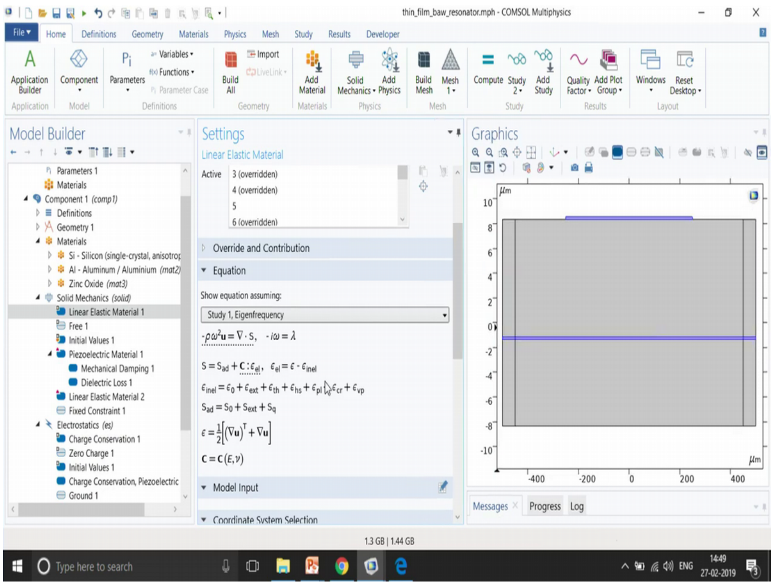
There could be mechanical damping or some kind of dielectric losses because which is actually hampering the force that is going to happen.

(Refer Slide Time: 04:30)



So, such kind of losses could also be taken into consideration. The most important part of what COMSOL has to offer it shows you all the equation. For example, if you go to electrostatics you know you are going to solve a Poisson's equation.

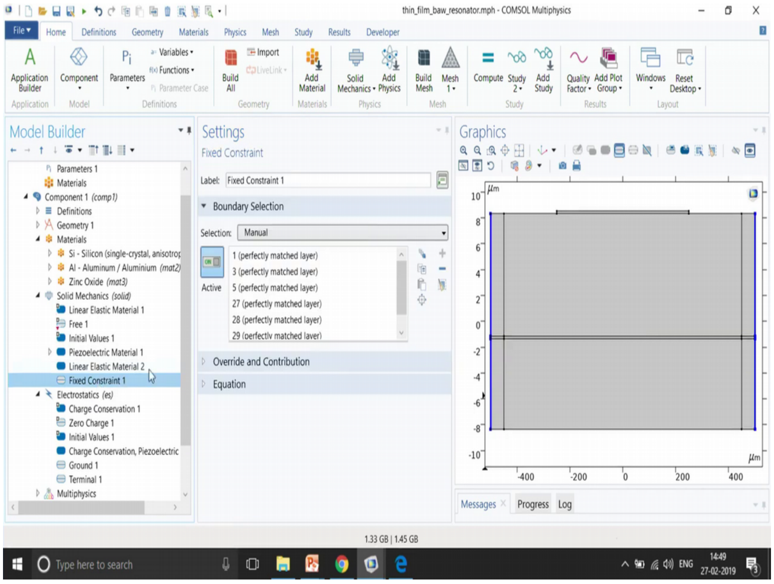
(Refer Slide Time: 04:57)



So, it actually shows the equation. For example, in linear structure linear elastic material it shows you what kind of equations you are going to solve for. This is the most important part of COMSOL that it is a very open software. It does not hide you what is it solving for.

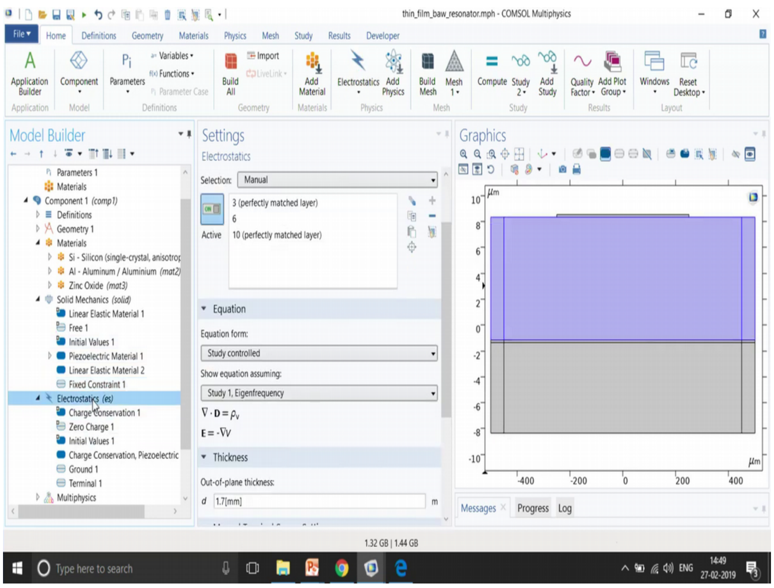
In piezoelectric material also you will see all the equations are available over here and the dotted part that, you can see over here is actually what you are giving through this particular setting node from this particular setting node. The dotted part is the what you are giving from the this particular setting mode and, along with the piezoelectric material modeling the linear elastic material for silicon bottom, which is an anisotropic material. These are the different types of material models which are available.

(Refer Slide Time: 05:42)



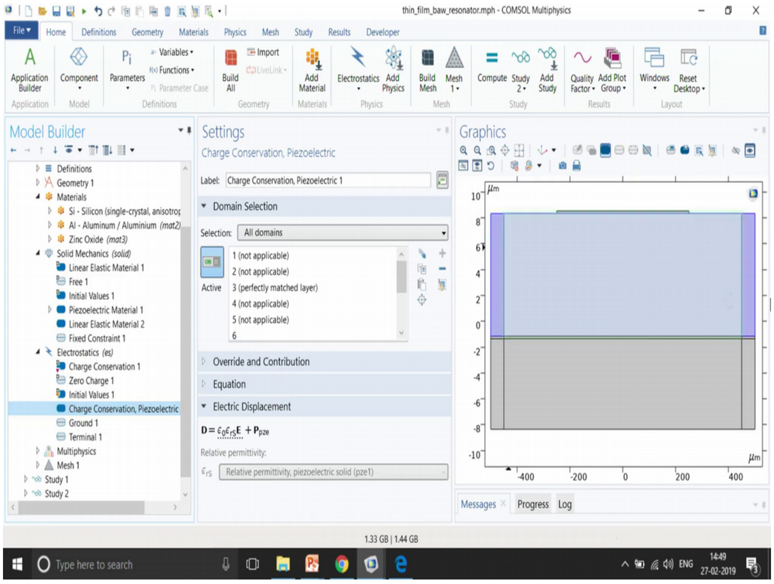
And then finally, we are doing fixed constant on the left and right side.

(Refer Slide Time: 05:46)



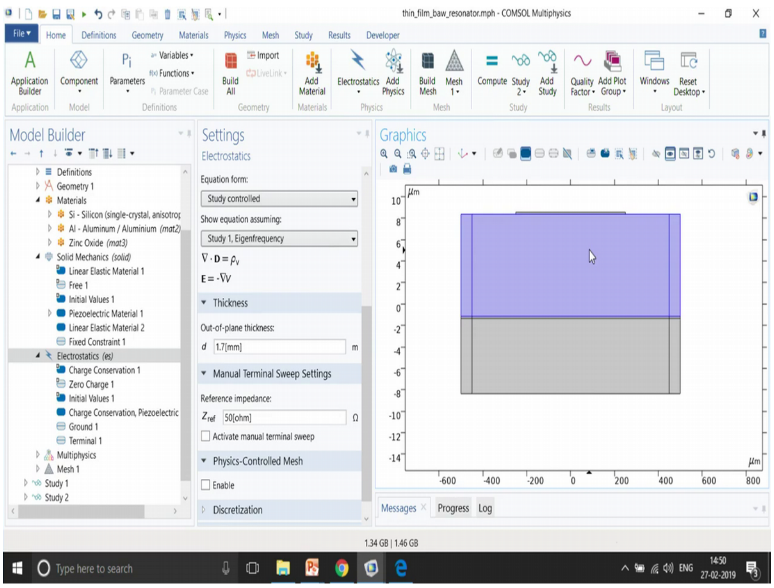
Then, the electrostatics part, where we are only solving for the top piezoelectric material;

(Refer Slide Time: 05:52)



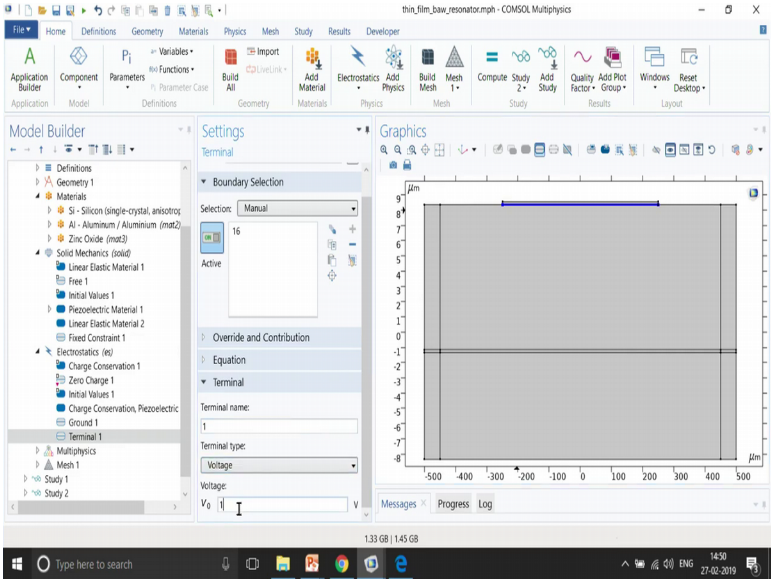
And, we are again giving a piezoelectric material model for the top part we are giving a terminal boundary condition.

(Refer Slide Time: 05:56)



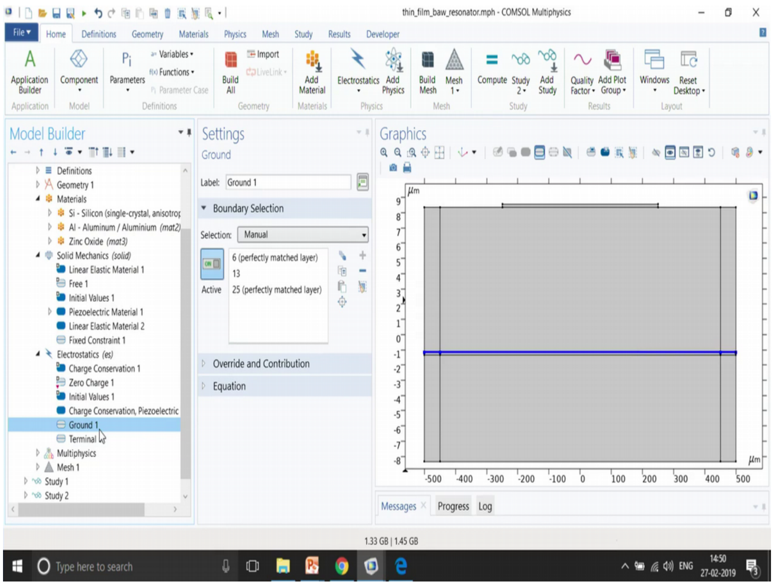
So, here it is important to know that we have not selected the aluminum part within the electrostatics. So, this is the aluminum part, but we have not selected over here because we know that the potential within this aluminum domain would be same. They would the electric field within this aluminum domain is going to be 0.

(Refer Slide Time: 06:20)



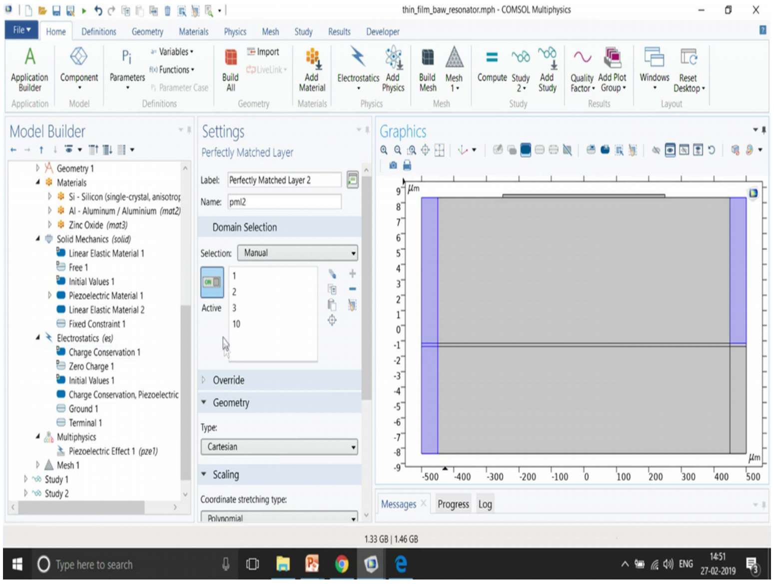
So, we use only boundary condition on the edges of your aluminium plates. So, here you can see in the bottom plate, we have given at terminal boundary condition with a voltage of 1 volt.

(Refer Slide Time: 06:31)



And, we have used a ground boundary condition on the bottom of the zinc oxide ok. And, then finally, we have used a multi physics node which actually couples the solid mechanics with the electrostatics ok. What I was telling you before is the use of PMLs.

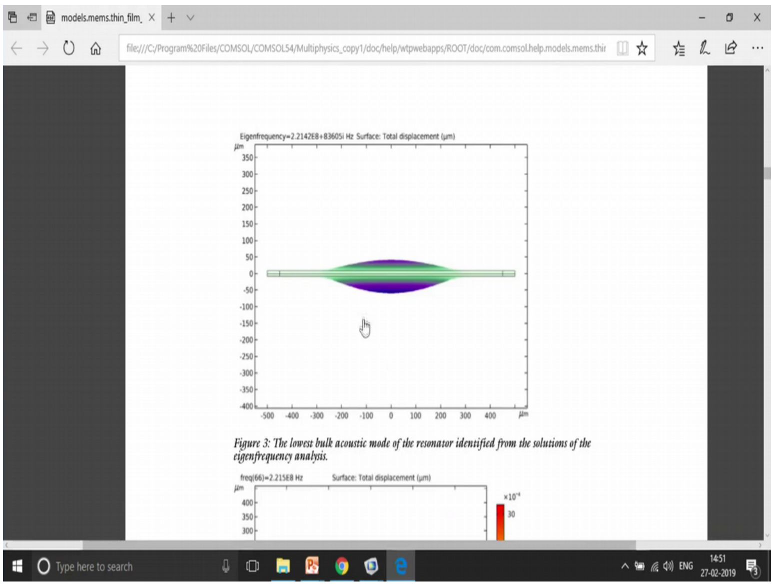
(Refer Slide Time: 07:00)



So, over here if you want to use the PMLs you can use it just right click on definitions and then you go to perfectly matched layers, and then apply PMLs over here right. So, this is how you can actually give the PMLs ok.

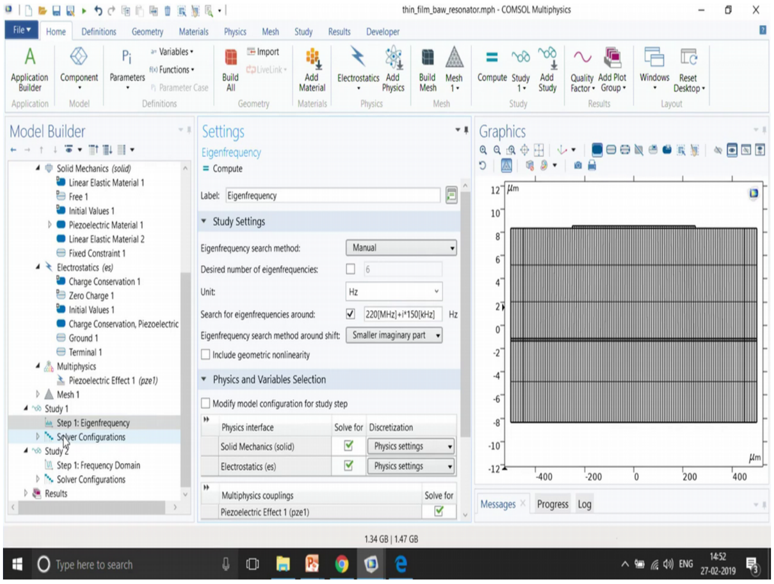
So, let me go and jump to the results part.

(Refer Slide Time: 07:18)



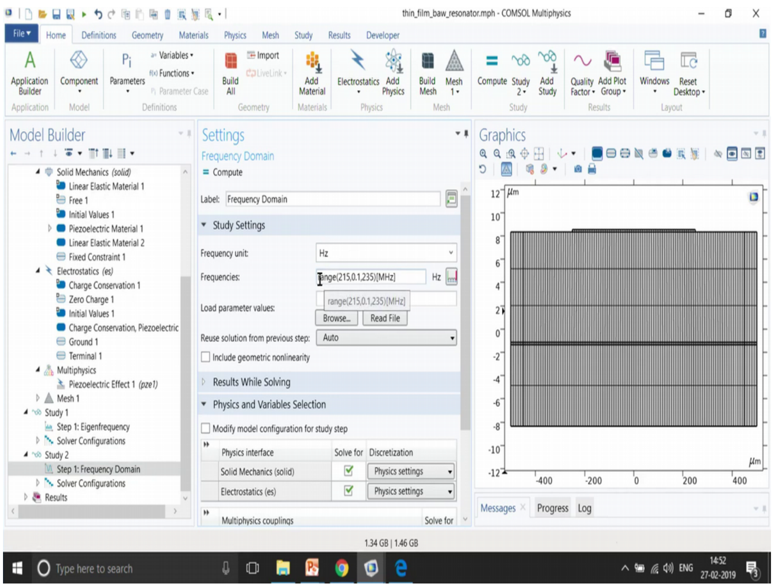
So, this is talks about the different acoustic modes ok. So, before results I need to show you what kind of analysis was performed?

(Refer Slide Time: 07:29)



Before analysis the mesh part so, in the mesh you can see a map mesh has been performed over here, very simple map mesh. If you want you can go with a physics control mesh also, but the map mesh will is more structured and in the first thing that you do again do is a Eigen frequency analysis that will tells you of different modes that could exist in your bulk acoustic wave. So, you have both the real and the imaginary part over here.

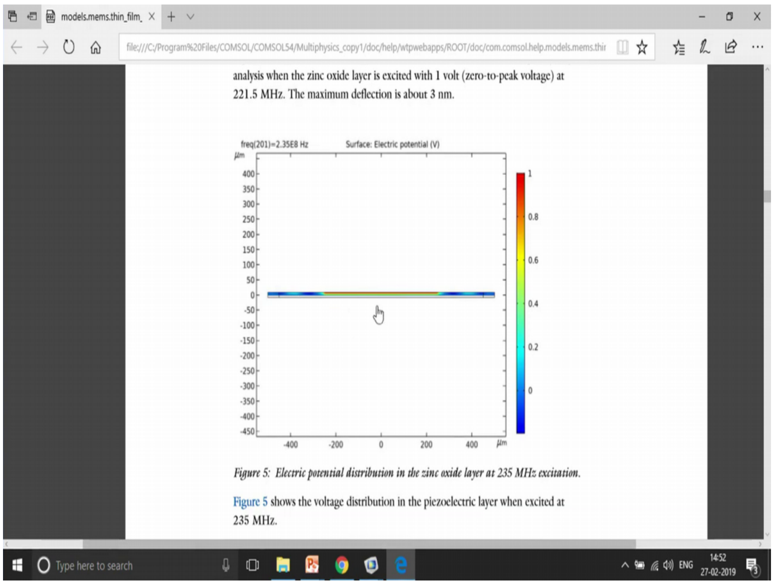
(Refer Slide Time: 08:03)



And, once you know that your Eigen frequency is around 220 megahertz you then do a frequency domain sweep. So, you sweep the frequencies from 512 megahertz 25 sorry 215 megahertz to 235 megahertz is tip of 0.1, and then see as parameters if you want you can see the ad admittance plot, all those things you can actually see.

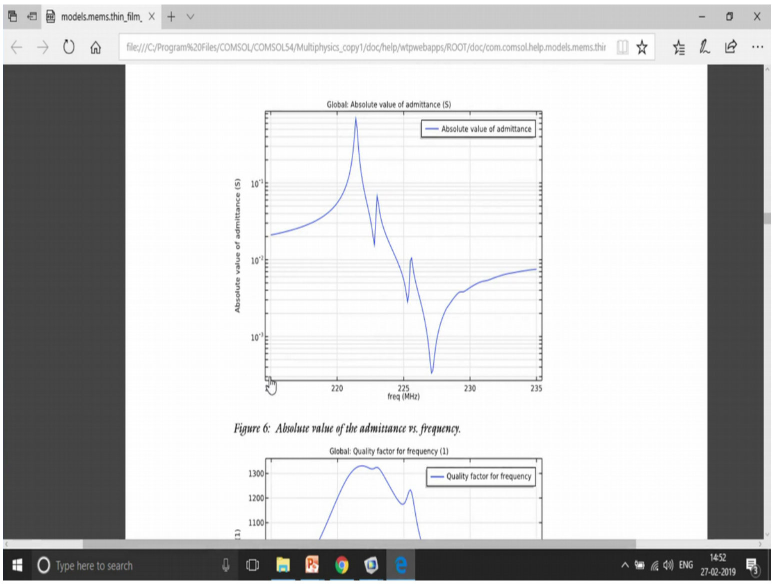
So, the results part this is not solved. So, let me just go to my documentation and then show you the results. So, this is the different resonant modes of your bulk acoustic waves.

(Refer Slide Time: 08:36)



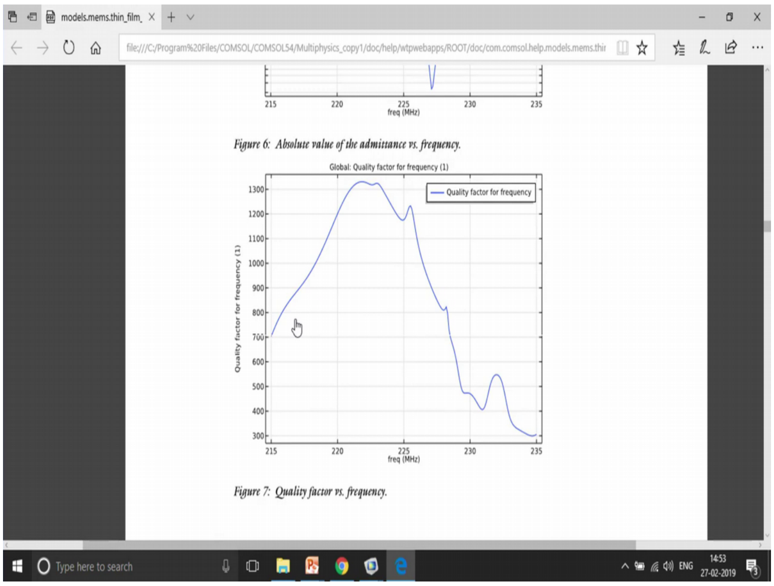
And, this is the talk talks about the potential drop.

(Refer Slide Time: 08:40)



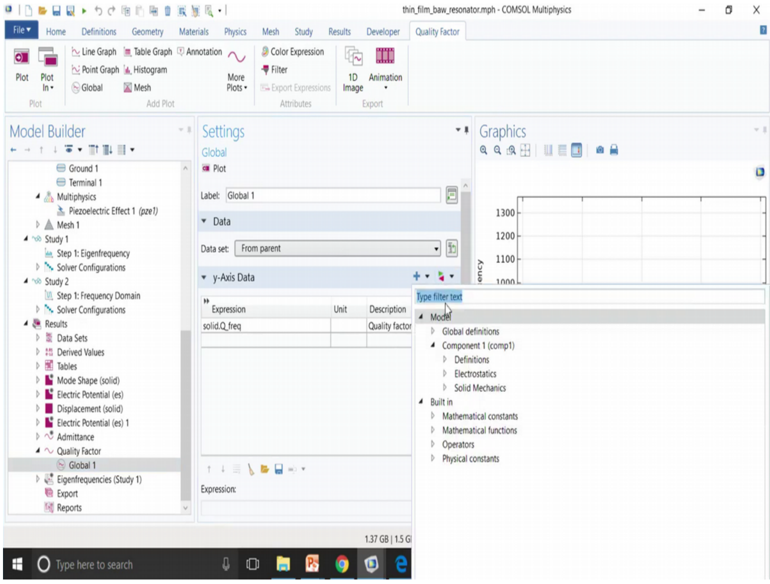
And, then this one talks about the admittance versus frequency. So, here we do a sweep of the frequencies from 215 to 235 and how does the admittance look with the sweep of the frequency.

(Refer Slide Time: 08:53)



The quality factor which tells you about the how sensitive your system is, can also be calculated with a direct form formula.

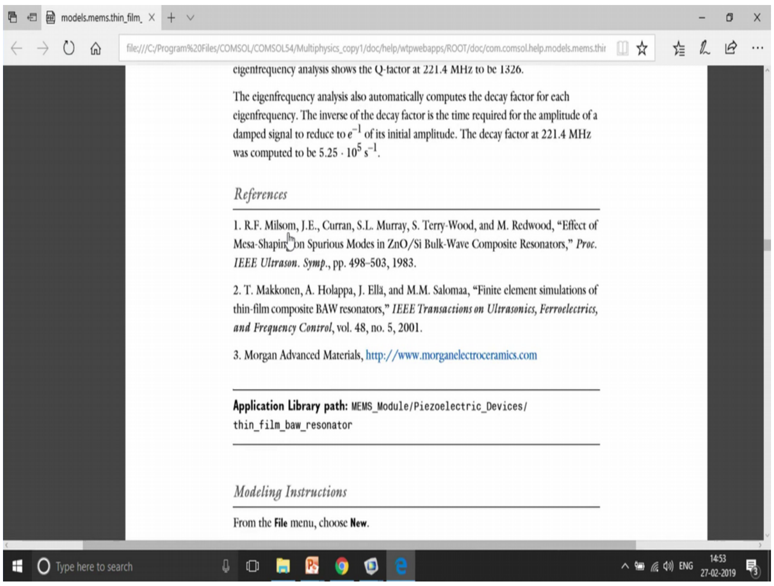
(Refer Slide Time: 09:06)



So, here with quality factor this is an internally defined variable. So, you do not need to search for that. So, you just search for the expression and this right quality factor for that you need to choose this thing it is q ok.

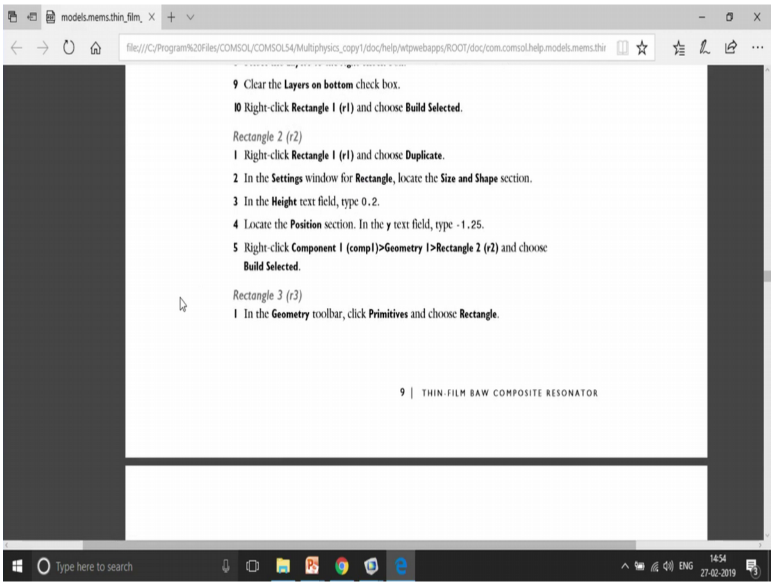
So, once you solve it at that time you will get. It is not search over here the quality factor and this variable will automatic come into the picture.

(Refer Slide Time: 09:41)



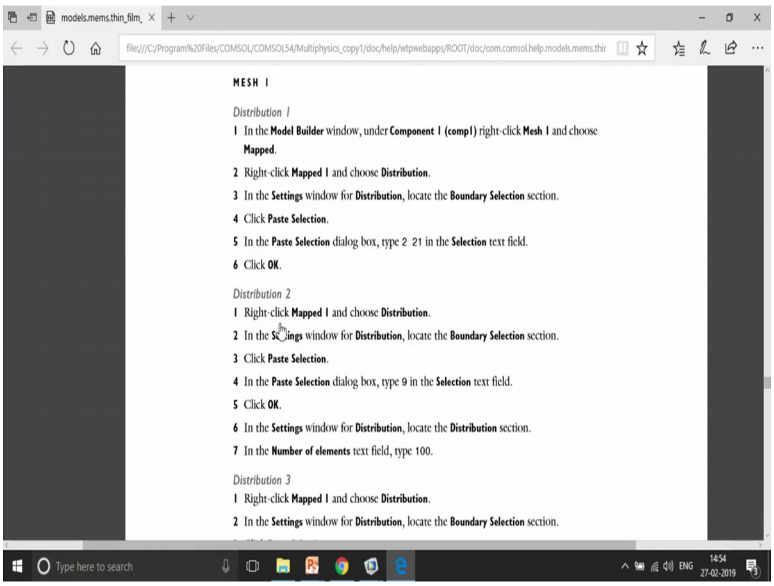
So, here the quality factor and again there are differences with which the results, the model has been compared with. So, I would strongly recommend you that once you do this simulation you also go through this references to know, how this module was set up and what are the material properties in why those material properties more importantly and then you have a step by step process to make the whole model.

(Refer Slide Time: 10:06)

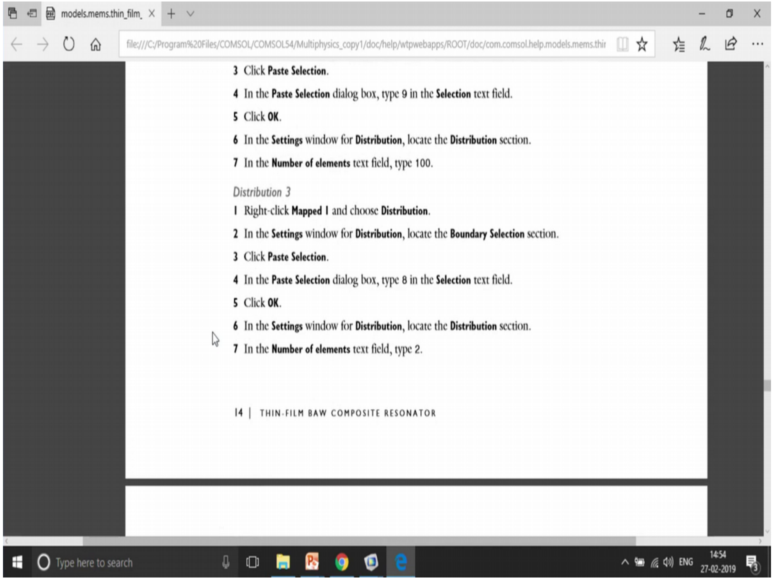


So, example you are looking to make a bulk acoustic wave then I will again strongly recommend that you do this model from scratch and then see if you are getting this results or not right and then only you go for your own design.

(Refer Slide Time: 10:22)

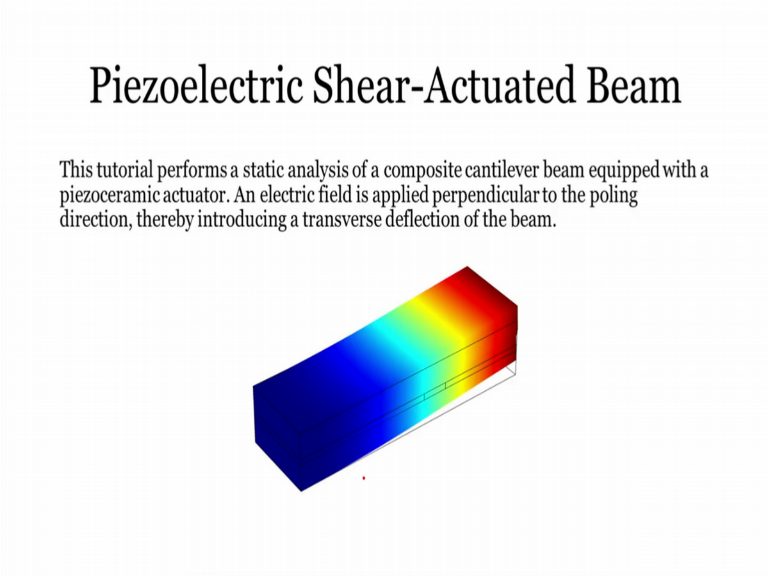


(Refer Slide Time: 10:23)



Once you are done with this particular example model then you can change this example model based upon your requirement and then see the admittance plot, the S-parameter plot and other analysis ok.

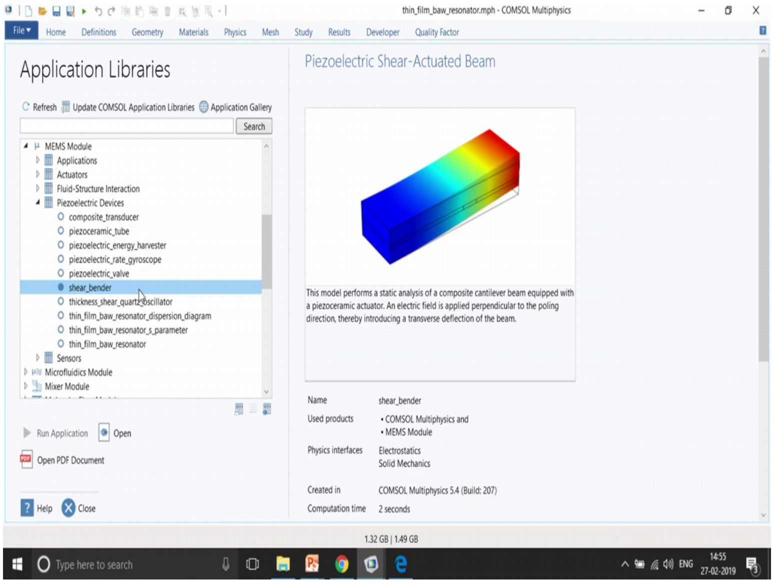
(Refer Slide Time: 10:37)



So, let us go ahead with one more example of piezoelectric shear actuated beam. So, these in this example what we are going to do is, we are going to have one piezoelectric device in the middle, which is covered with foam on the left and right side and then we have two structures elastic structure elastic models on the top and bottom and we are going to impose a particular voltage which is going to create a particular stress and that particular stress is going to deform the beam.

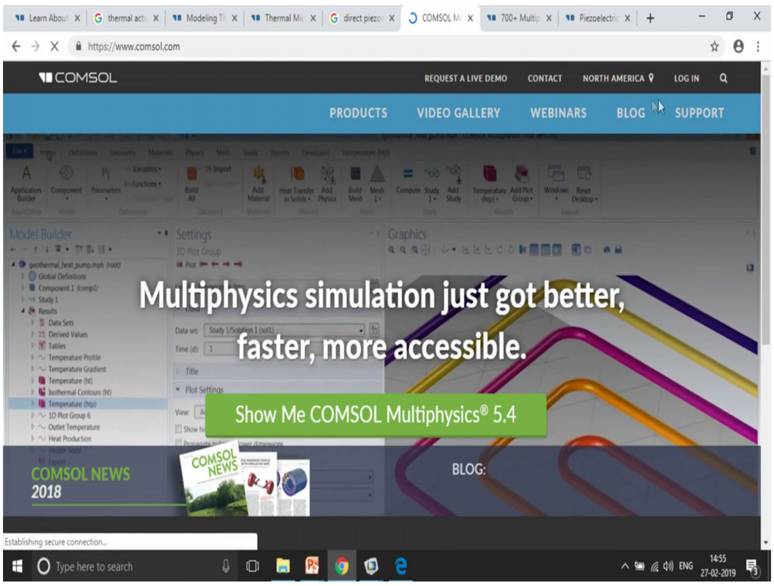
So, let us go open the model.

(Refer Slide Time: 11:15)



So, I just go to COMSOL, I go to File, Application Library and over here I search for shear bender. So, this is example model this is again a very example a very nice example model because it talks about the poling direction being different as what is given from the by default.

(Refer Slide Time: 11:37)

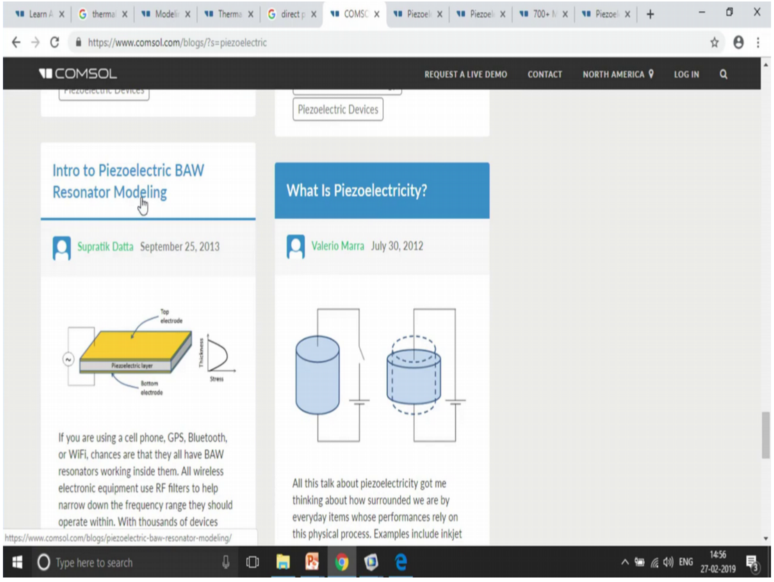


If you want to know about the poling poling directions, so, I would recommend you to go to the COMSOL block. So, this is a COMSOL block. So, if you go to COMSOL dot com and then go for the Blogs.

(Refer Slide Time: 11:46)

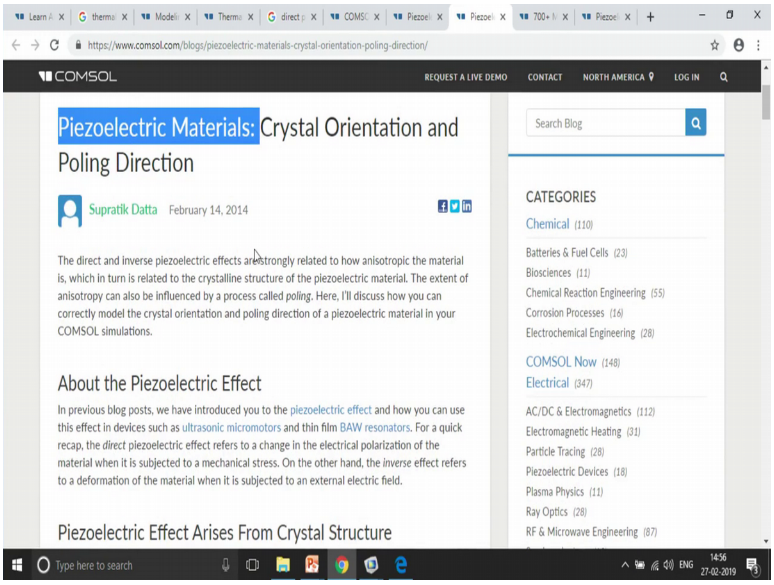
Yeah. So, we have many models on piezoelectric devices the one that I am telling you about is yeah one of example is understanding the standard. So, there are many different standards of piezoelectric materials available. But, which standard should you actually go through? What are the different poling direction what is the basics of poling direction?

(Refer Slide Time: 12:07)



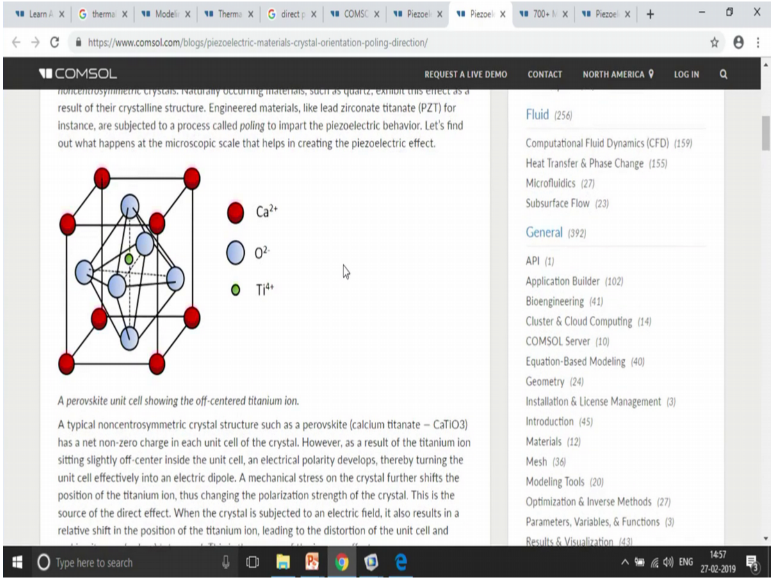
This also models blog for resonator model right. So, you can also see this blogs because blogs are very important because they explain ways in a very simple manner, how the physics has been captured and how you can actually do it do a modeling out of it. So, again recommend you to go through the blogs.

(Refer Slide Time: 12:27)



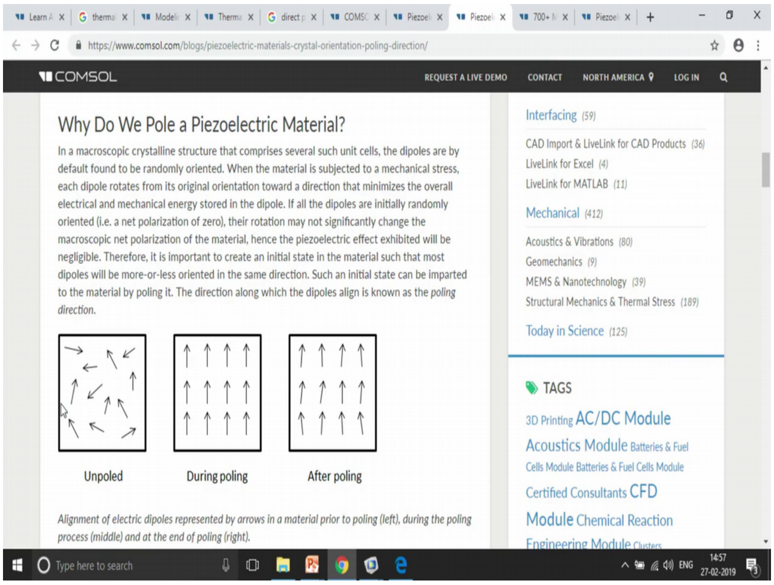
So, I was talking about the poling direction. So, a piezoelectric material along with this orientation and poling direction.

(Refer Slide Time: 12:35)



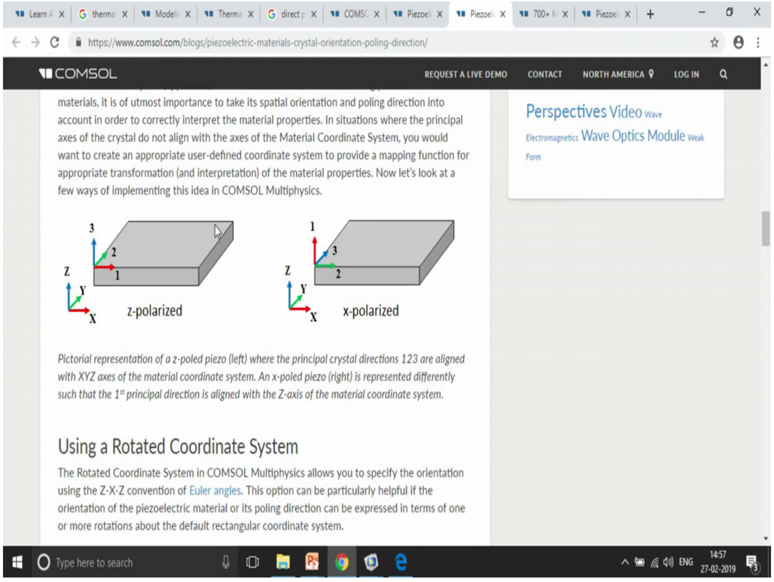
So, any particular material will have it is own poling direction based upon where these atom which is defining that poling direction is associated with.

(Refer Slide Time: 12:50)



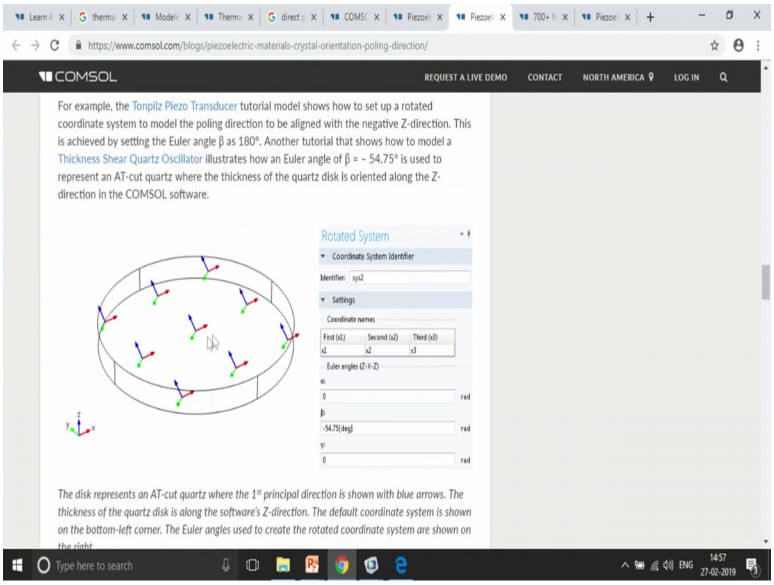
And, once you give a particular external electrical field you will see the dipoles to be arranged in a particular direction quite. So, during the poling it so, it is a particular direction that is along the electrical field.

(Refer Slide Time: 13:03)



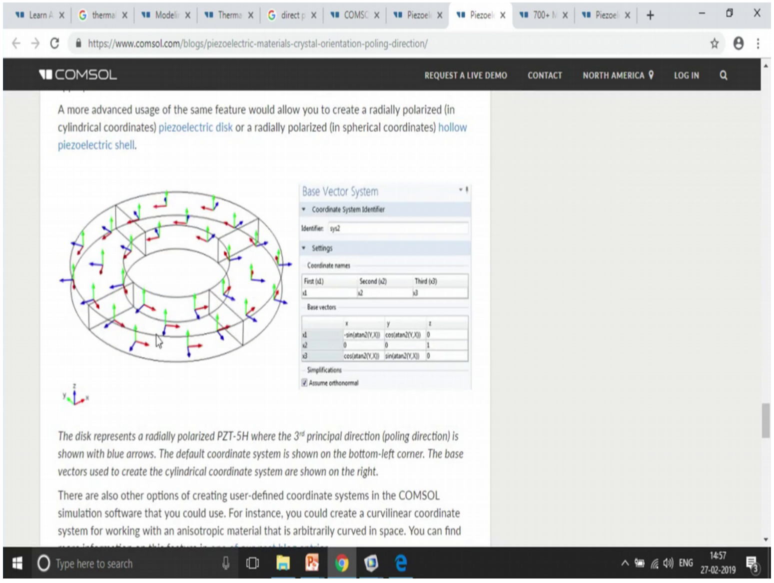
But, if you cut a particular crystal in a particular angle then the poling directions could be updated right.

(Refer Slide Time: 13:12)



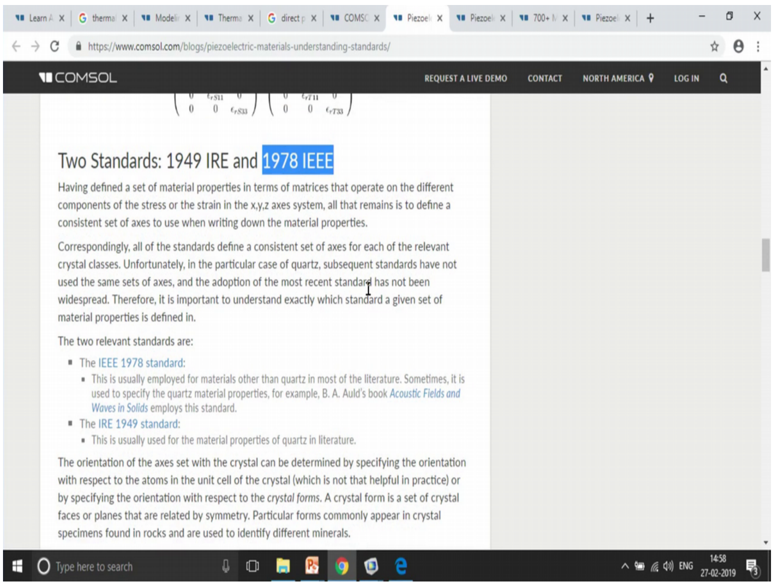
So, if you want to give a particular angle to the poling directions you can actually use a rotated systems right with the different different angles that you can see over here different material properties.

(Refer Slide Time: 13:24)



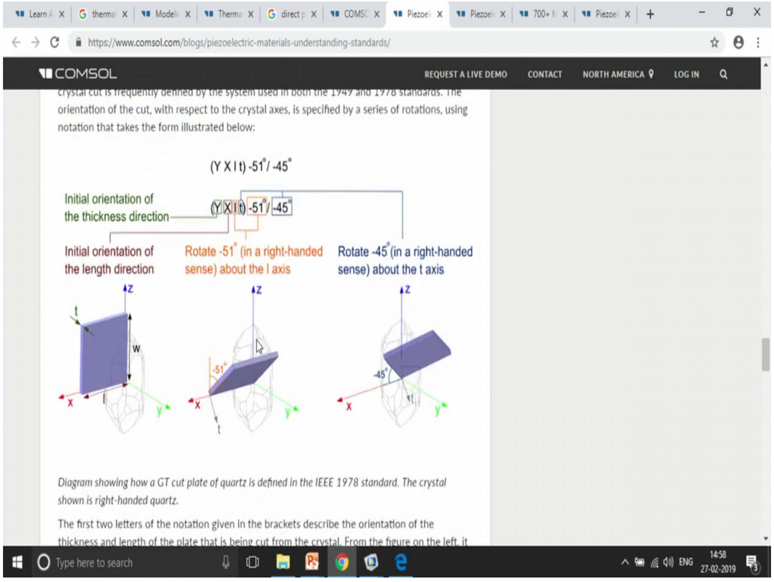
If you want to use out which is radially outwards alright, poling direction which is radially outwards you can define those particular types of co-ordinate systems and then assign the piezoelectric material based upon this particular co-ordinate systems, radially outwards.

(Refer Slide Time: 13:47)



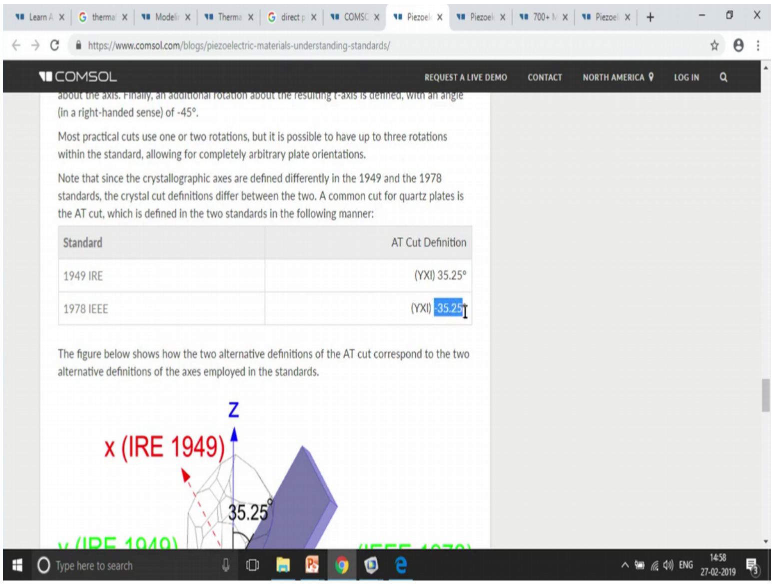
Then, we have different standards of piezoelectric materials right. So, basically 1945 IRE and 1978 I triple E, these are two basic standards for piezoelectric materials and those also have a particular kinds of poling directions right.

(Refer Slide Time: 14:00)



With a particular cut which is defined based upon their the specification numbers that they have alright.

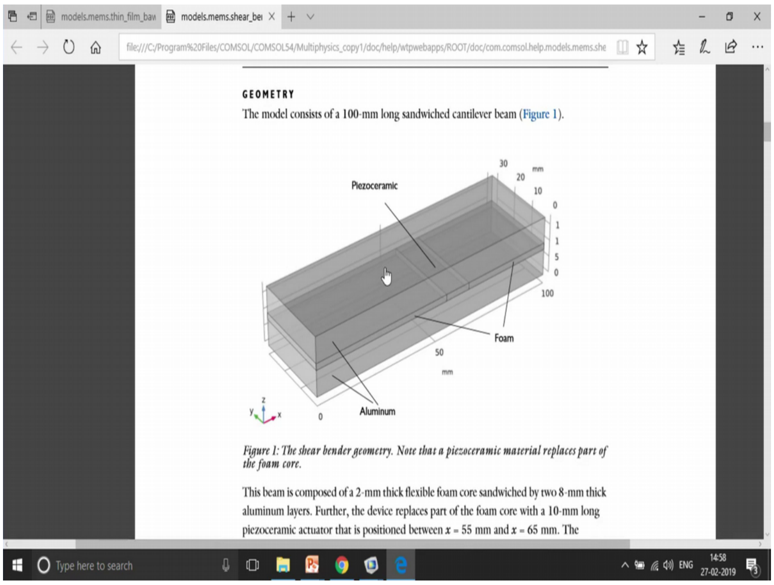
(Refer Slide Time: 14:08)



So, those kind of small small differences we can actually easily model take into account while modeling in COMSOL.

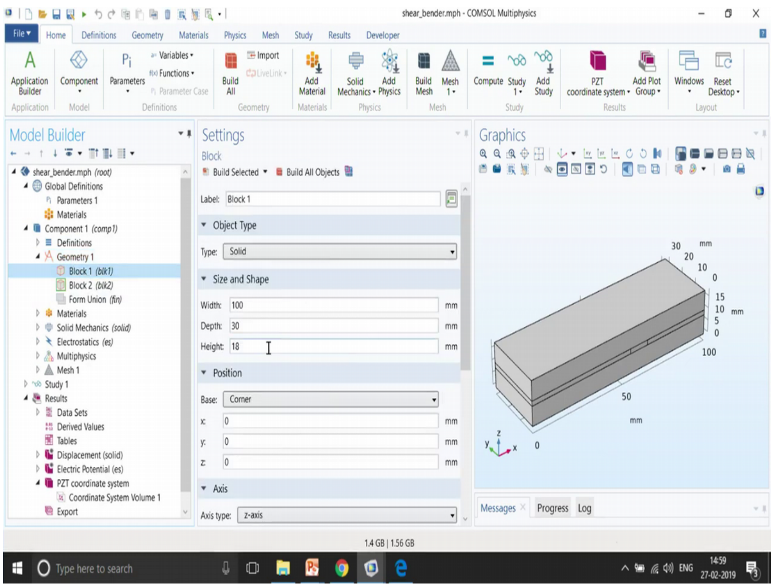
So, let me just go ahead and open this particular model.

(Refer Slide Time: 14:27)



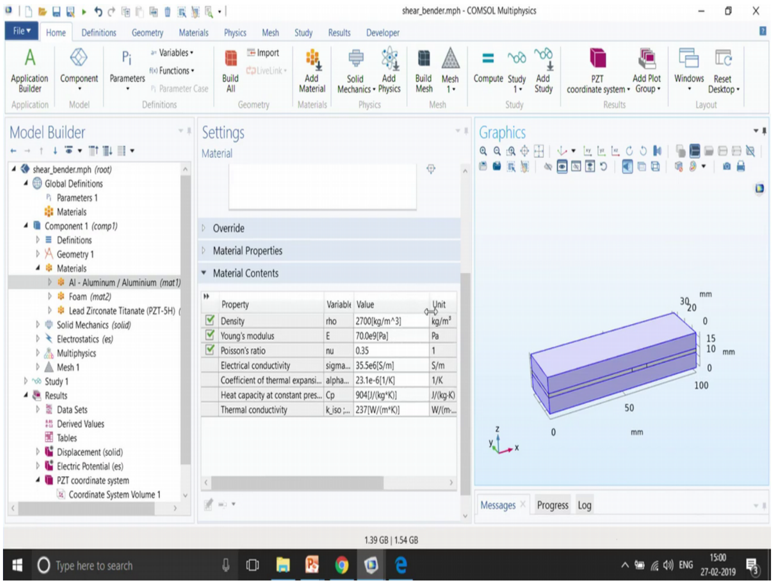
Again, you can open the documentation to see how the physics set-up has been performed. So, over here you can see that we have a piezoceramic domain in the top and we have the same sorry piezoceramic is the piezoelectric material and we have our aluminum plate in the top and in the bottom. And, and on the side ways of the peizoceramic material foam has been applied. So, on the left and right of the piezoelectric material foam has been applied right and then we give in the top and the bottom of the peizoceramic material we give a particular voltage to it. So, let me just go to COMSOL and open this particular model ok.

(Refer Slide Time: 15:11)



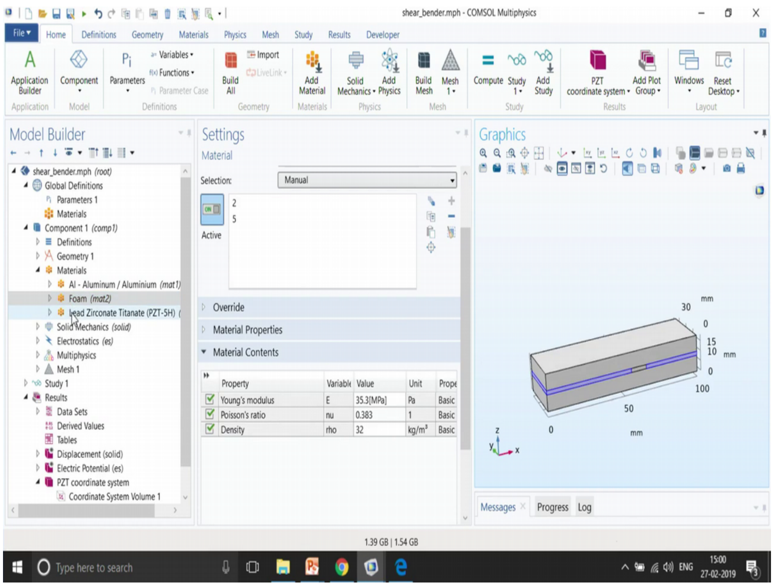
So, let me go from top to bottom. So, in the component section first thing is to make the geometry. So, I just go ahead make the blogs and make the geometry, I first make a block and they are introduced the centre domain using layers over here.

(Refer Slide Time: 15:32)



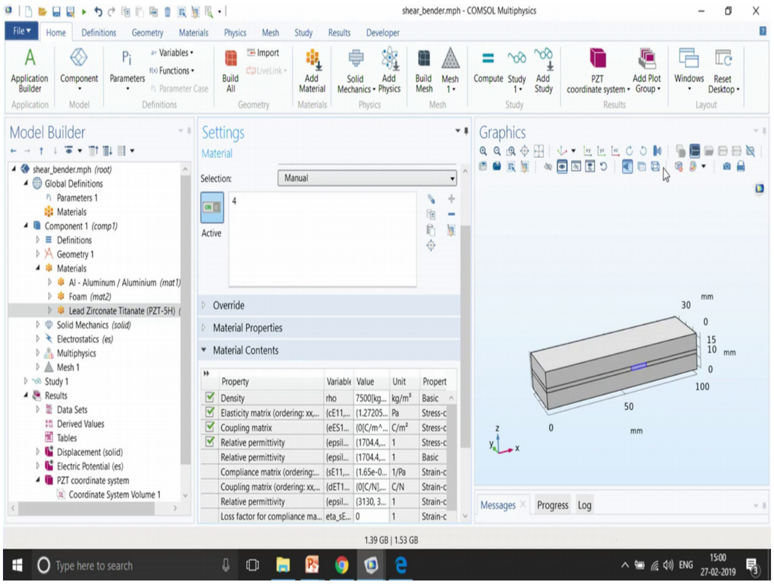
And, then I assign the material properties most important, aluminum domain in the top and bottom. You can see the blue part the top and bottom is aluminum domain and here you assign it as a scalar material properties ok.

(Refer Slide Time: 15:50)



Then, we have foam which is on the left and right that you can see in the blue part in the screen and this again is a scalar material properties where you can see.

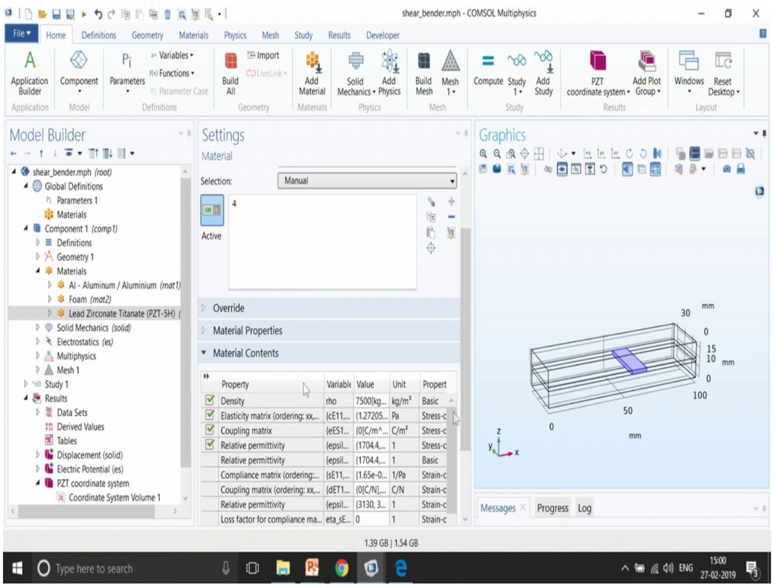
(Refer Slide Time: 15:59)



And, then finally, the PZT-5H and here we give a tensile material properties as you can see over here the blue part.

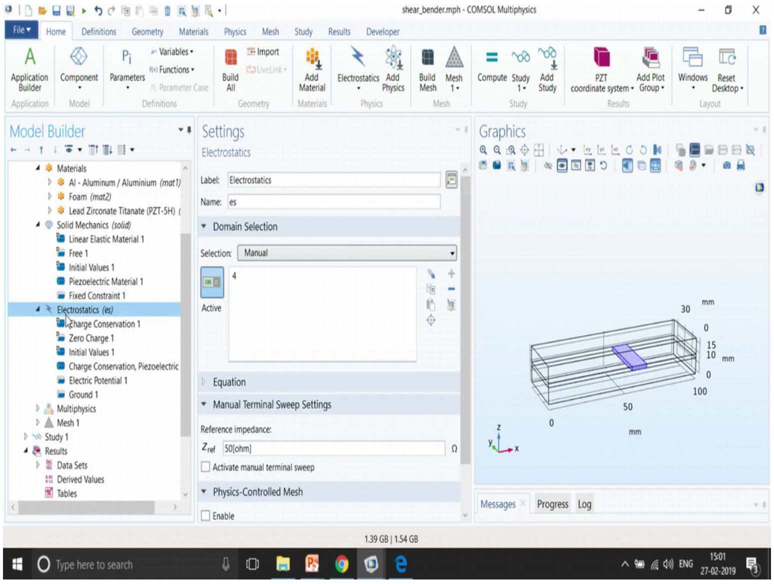
If you want to see through the domain, you can anyway enable the wire frame rendering.

(Refer Slide Time: 16:12)



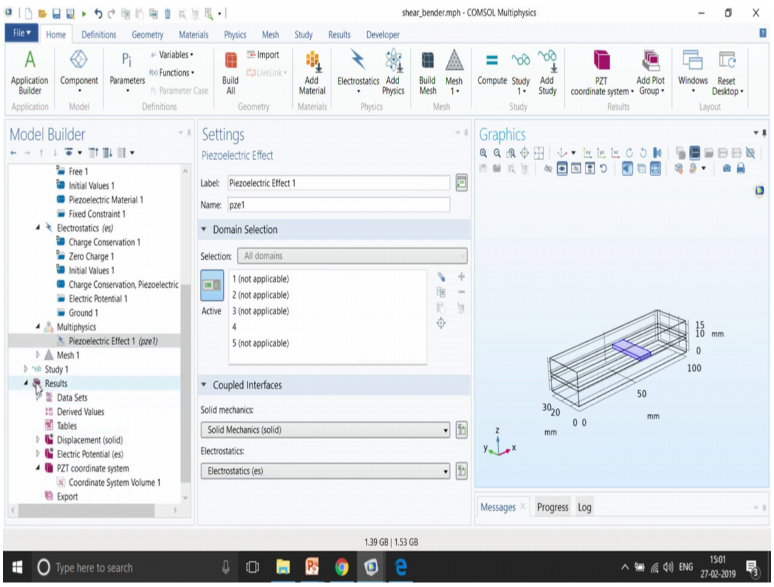
So, you can see you can see it through.

(Refer Slide Time: 16:17)



Next part is a solid mechanics part and over here again we use piezoelectric material over here the way that we define earlier we give this works as a actuator beam that is why we give a fix constant on the left side ok. So, it is going to act as a beam. So, if there is going to be some deformation it is going to deform in the up or the bottom.

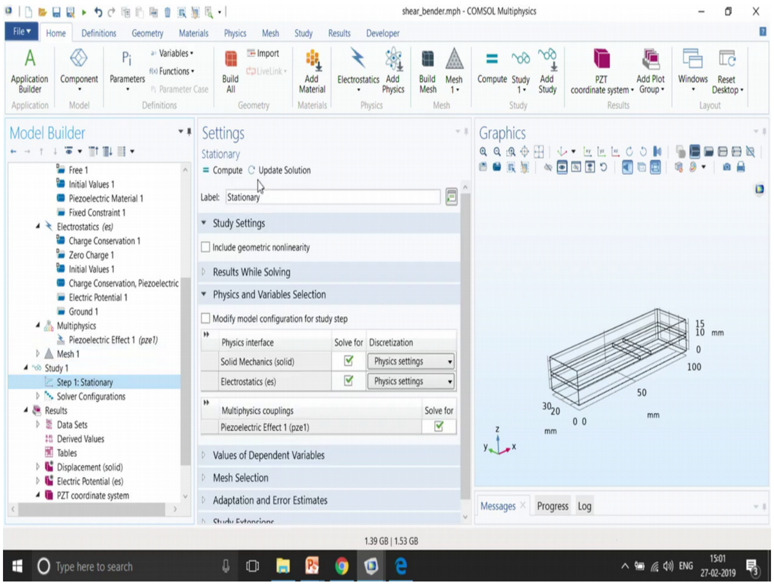
(Refer Slide Time: 16:41)



Then, we have electrostatic again we use the material model piezoelectric material model and we give the electric potential in the bottom. So, you can see over here in the bottom part I have given the piezoelectric material property and then in the top part I have given a ground boundary condition. And, then finally, to couple both of them I am using piezoelectric coupling.

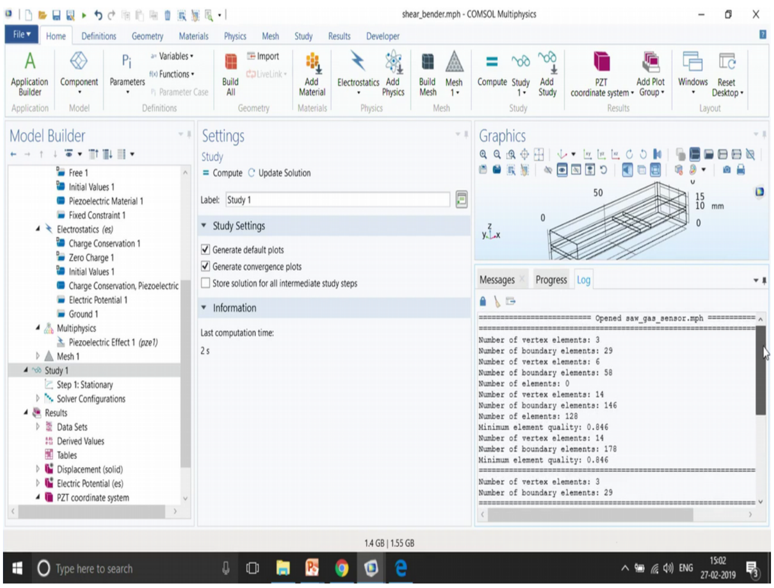
So, this actually couples the solid mechanics with the electrostatics.

(Refer Slide Time: 17:12)



And, then I am using a kind of a stationary study.

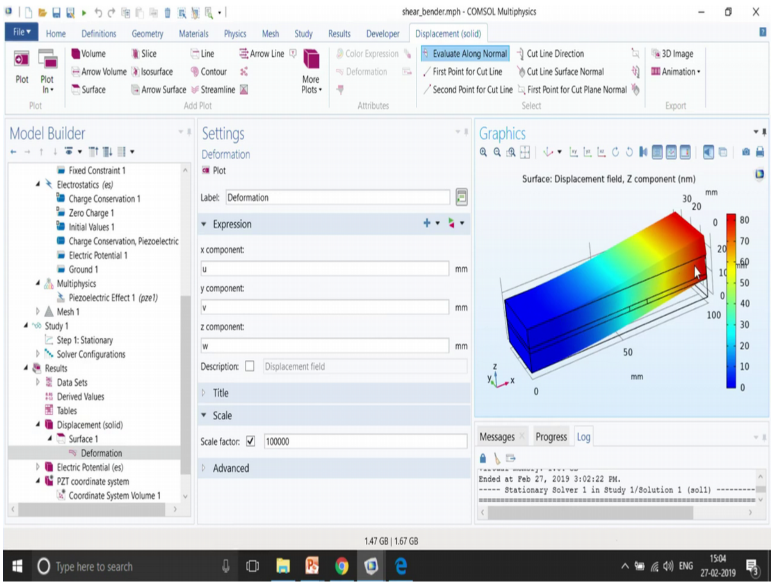
(Refer Slide Time: 17:14)



So, this is going to take only 2 minutes alright. So, you can see how much time it will going to take hence always good to see how much time it is going to take because some of the models are large. For example, the example that we are going to talk about in sometime is an example of accelerometer. It takes little bit more time any addition to the more time it takes you more RAM right. So, maybe your system is not having that much of RAM. So, it always good to see how much time it is going to take.

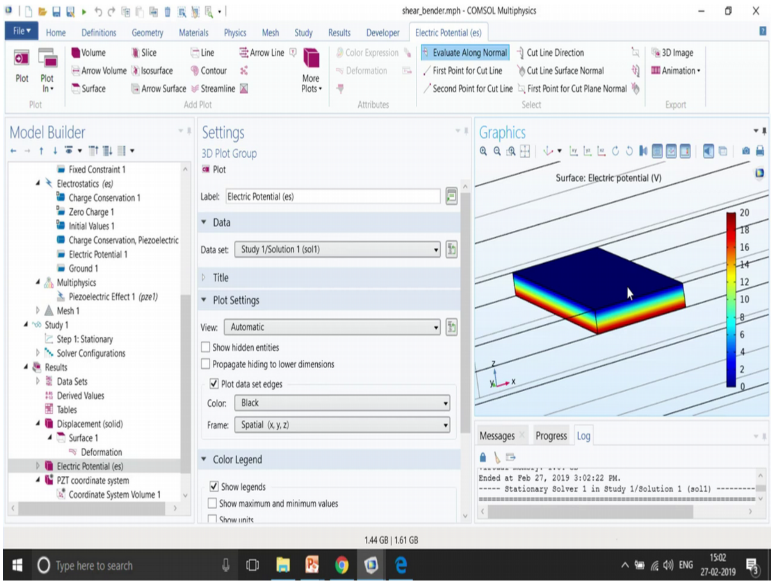
And, if you want to see how much time it RAM it takes I think you should be available in the log values also. So, here is also can see how much it may be showing how much RAM, it took to solve this particular model.

(Refer Slide Time: 18:00)



So, let me just go to compute. So, now, we can see that the displacement which is in order of nanometers. So, what you see right now is an exaggerated view of what is happening within it.

(Refer Slide Time: 18:17)

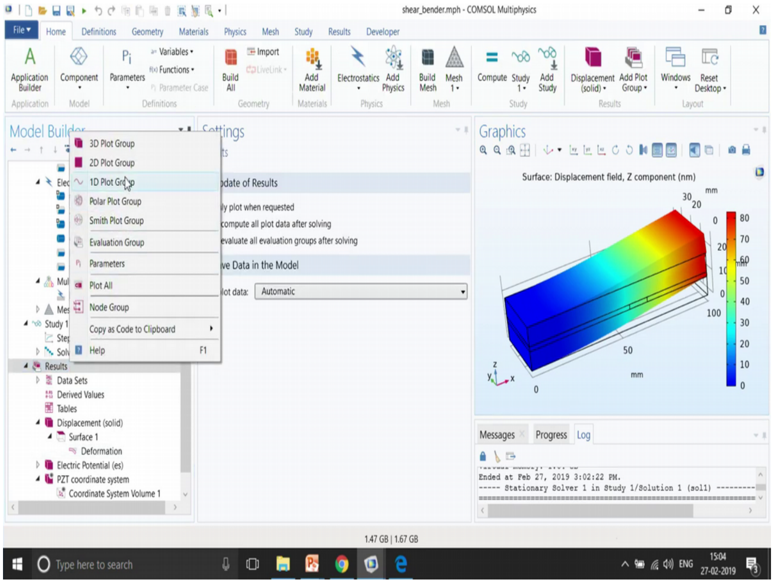


The first thing is the electric potential. So, we give a high potential in the bottom and ground it in the top right. So, you can see a nice deep or deep in through the from the bottom to the top, a linear deep.

And, then in the displacement because of it there is going to be some stresses and because the stress is it is going to deform. So, in this case is as deform in the top part. So, it is moving top right moving upwards. If you are make the potential in the opposite direction that is high potential in the top and ground in the bottom then the deformation will happen in the downward direction.

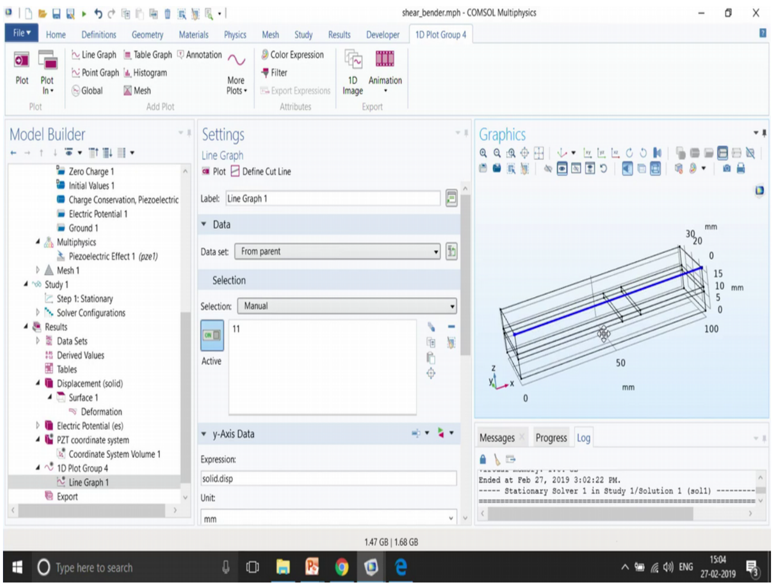
Over here, as I was saying that this is a scaled version; so, if you want to see the actual deformation then you to go within this surface 1 to the deformation that we can see over in the screen and make it as 1 and then click on Plot. So, this is the actual deformation. Of course, in 90 nanometers in device of mm structure, you will not be able to see that much. So, even if you just zoom it you will not be able to see that much and that is why we have just scaled it to show you the effect of the deformation ok.

(Refer Slide Time: 19:43)



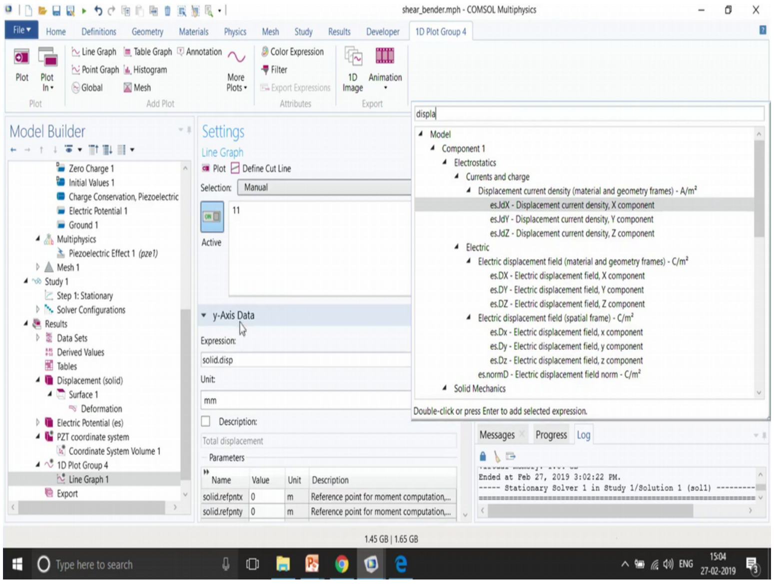
So, for example, now you want to see the deformation at this particular line that is also possible.

(Refer Slide Time: 19:48)



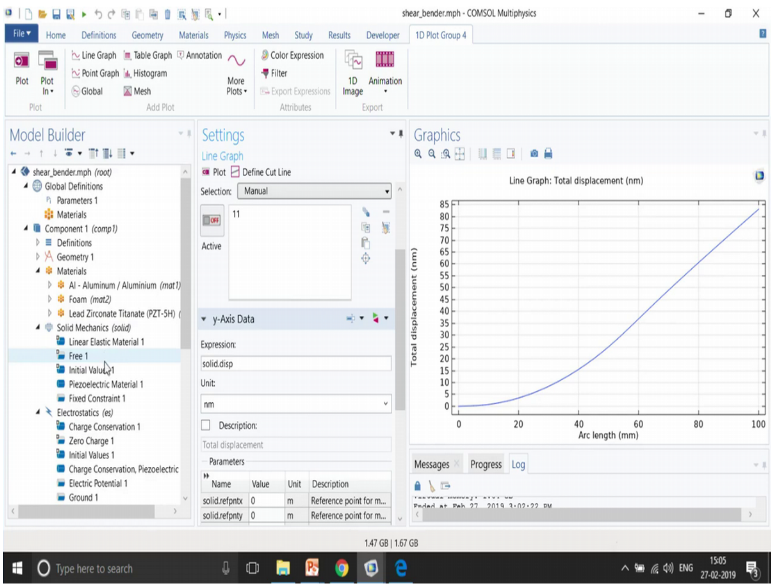
To do that just right click on the Results 1D Plot Group. Add 1D 1D Line Graph and choose which line you are talking about. So, I am talking about this particular line all right. So, by default this displacement variable has already come.

(Refer Slide Time: 20:02)



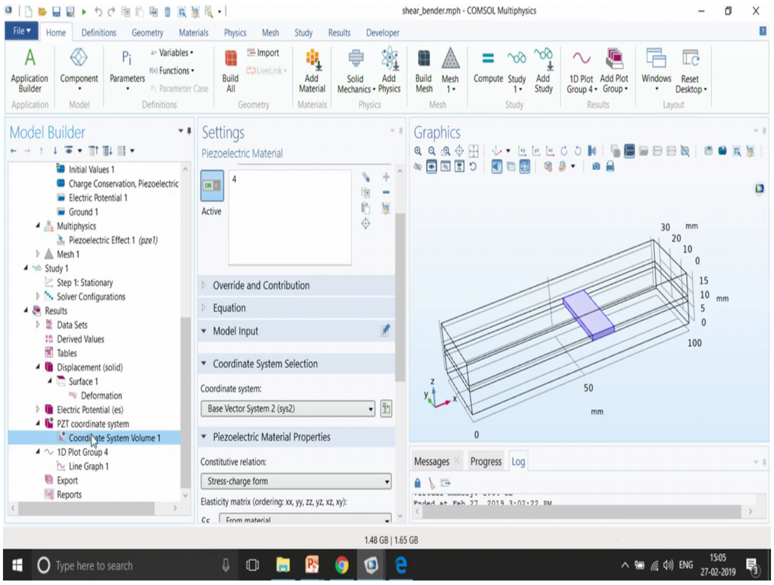
So, if I want to search I can also search over here. So, I go over here and the search for displacement and there are many other variables. So, right now I am looking for total displacement right. So, I just use the total displacement over here.

(Refer Slide Time: 20:20)



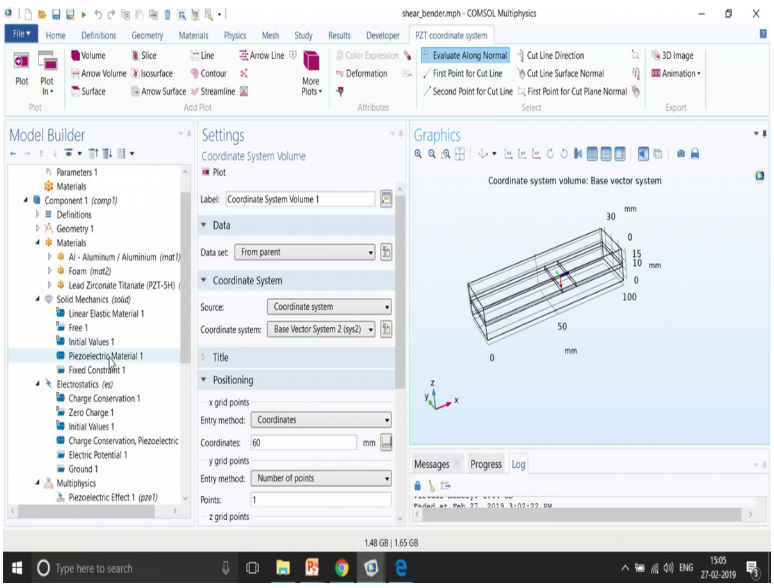
Just double click on Total Displacement and you will get solid at display over here and you can see the displacement. So, you can see from 0 to around 8.3 or around 83 micrometers is the displacement sorry, it is actually nanometer. So, it is in mm. So, let me just go ahead and give nanometers. So, around 83 nano meters is the maximum displacement along that particular line.

(Refer Slide Time: 20:52)



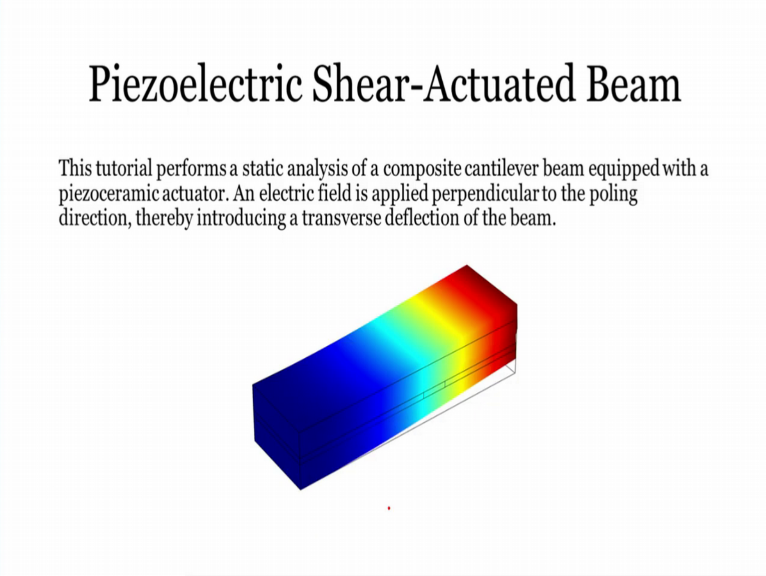
Finally, you can see that in this particular case we is very important example because in this piezoelectric we have used a different poling direction. So, by default you will see the poling direction of the any of the material properties is along z-direction.

(Refer Slide Time: 21:07)



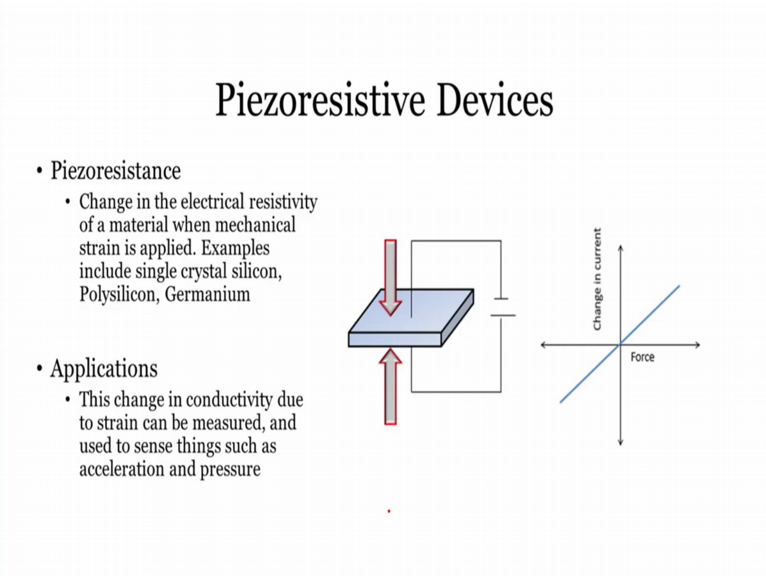
But, in this case you can see over here. It is along x-direction, the blue the blue line it is around x is a blue is the one is the poling direction and that is along the x direction. The way it was taken into consideration that is the poling direction of in x direction be use in definition, we use a base vector system and we define that particular poling direction over here in the form of base vectors.

(Refer Slide Time: 21:40)



Looks good. So, let us go ahead with a presentation. So, this is how the different types of piezoelectric materials were modeled.

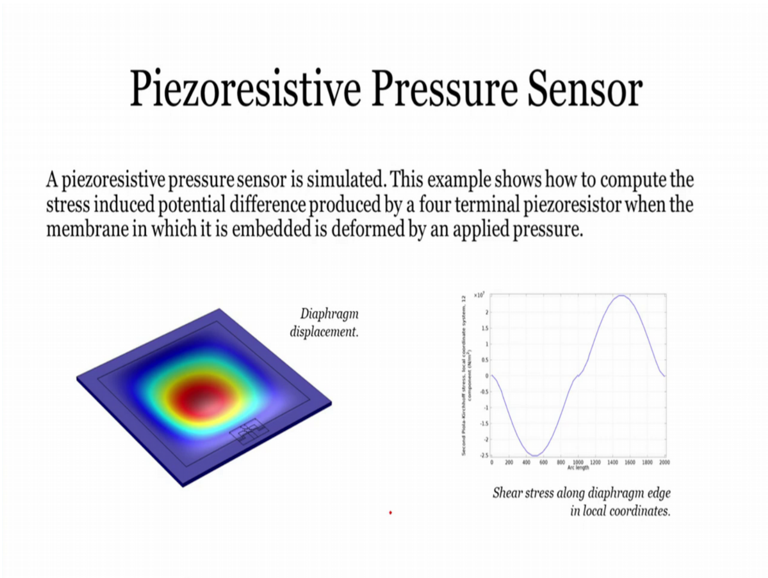
(Refer Slide Time: 21:45)



So, let us go ahead with the piezoresistive device. This is a very interesting type of a device because over here based upon the forces that we apply or this stresses that are going to develop, it is going to change the resistivity of that particular device and the way that you can take into account, the resistivity as a sensor is to understand how much current that is being withdrawn from that particular system.

So, in this particular example that we are going to talk about how the forces are being applied, because of forces how the resistance are going to change and because of the change in resistance, how the current is going to change.

(Refer Slide Time: 22:23)

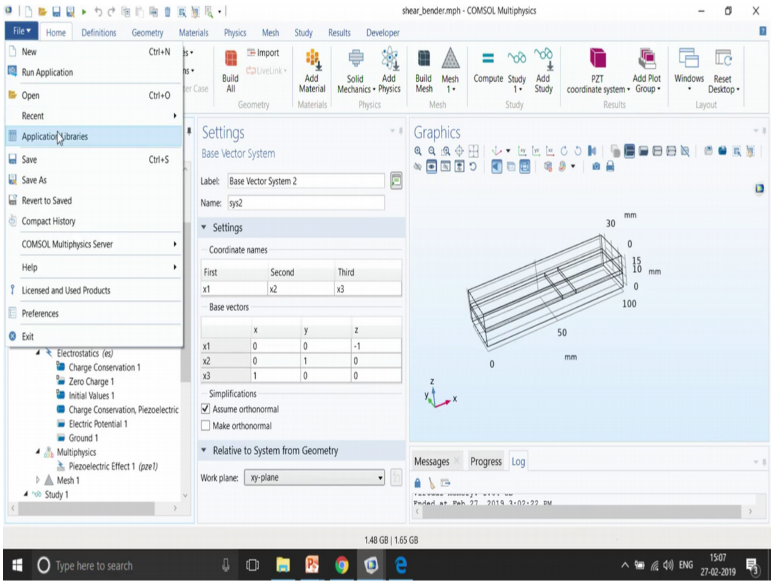


This is the first example that we will talk about. This is an example of a diaphragm that you can see over here and we apply a particular force on the top from here and we have a piezoelectric sorry piezoresistive material in this case over here with different types of doping.

So, n-doping and p-doping both we have over here and then we try to understand along particular arc in this case how the stresses have being developed. So, you can see both the negative and positive part of the stresses that is getting to picture and because of that you can understand how much current is going to be getting placed.

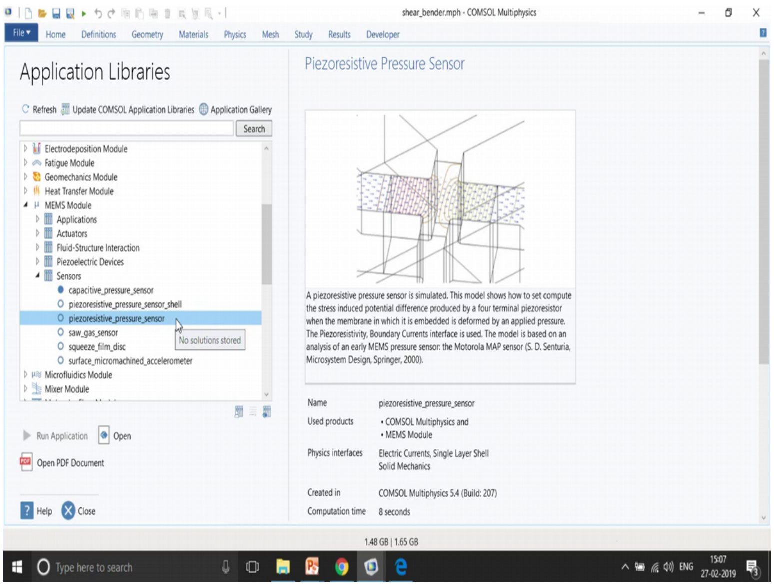
So, let me just go to COMSOL again.

(Refer Slide Time: 23:07)



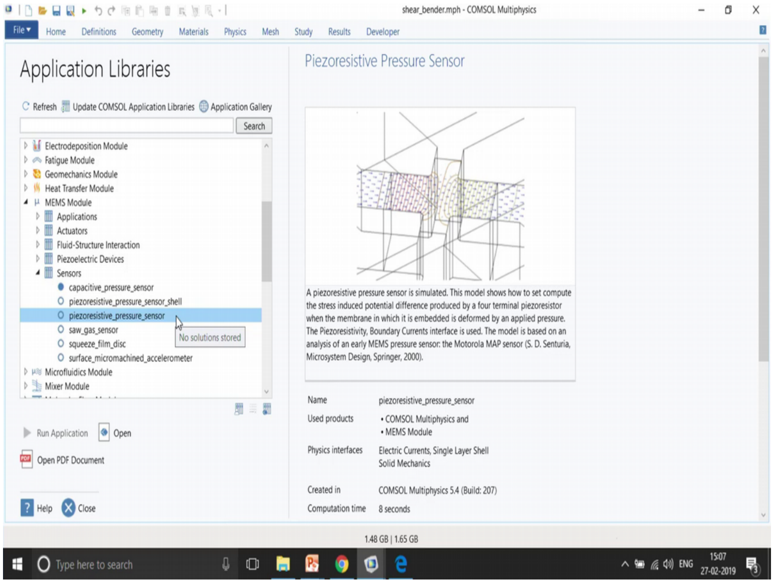
So, I go to File, Application Library and I search for Piezoresistor ok. So, it could be somewhere over here I guess yeah.

(Refer Slide Time: 23:10)



So, there are different ways of modeling again. One is a volumetric model that you can see right now.

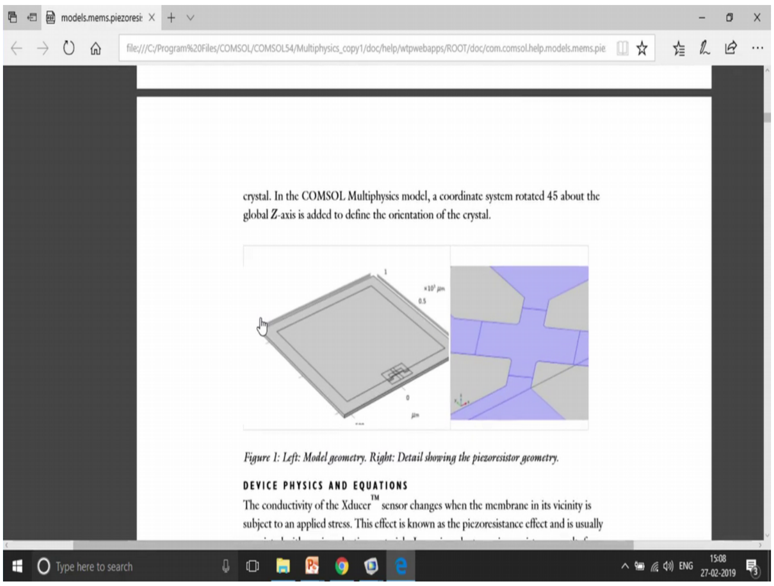
(Refer Slide Time: 23:34)



The another approach is a shell approach that is modeling only 2D structures. So, this is again different. So, it is kind of a approximation that we do to quickly get the results which is the results which are not that far away from the actual results.

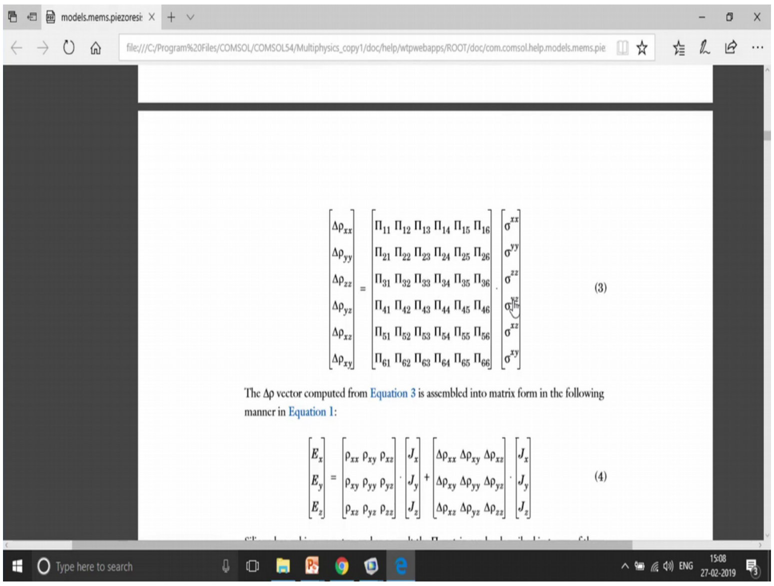
As of now let us go for piezoresistive pressure sensor. Let us open documentation.

(Refer Slide Time: 23:57)

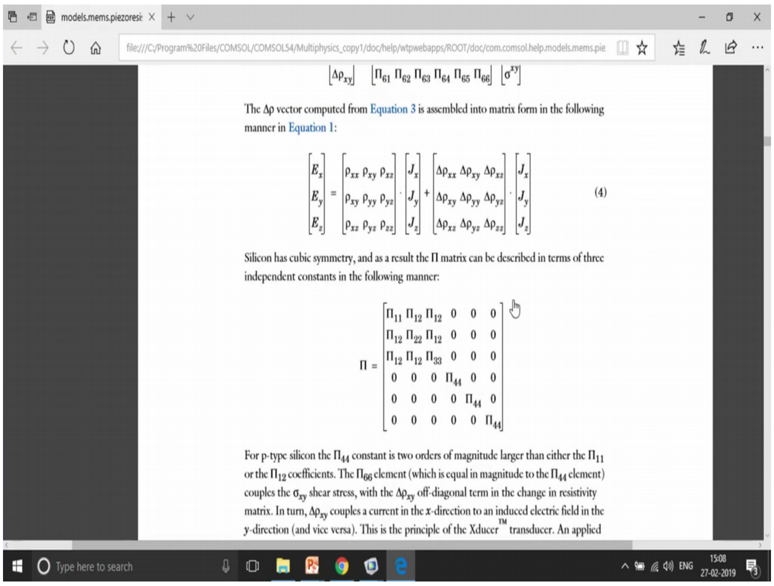


So, this is the actual structure that you can see over here. This is a diaphragm over here and then we have the piezoresistive structure over here which is kind of a Wheatstone bridge kind of a system, where we give a particular voltage on the one side and ground on the other side and the other two flanges, that you can see over here that is used for decreasing the error rates.

(Refer Slide Time: 24:22)

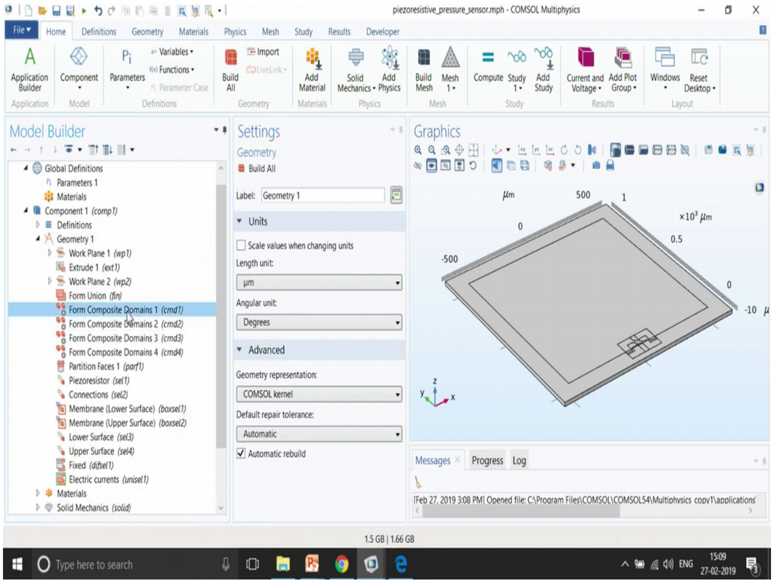


(Refer Slide Time: 24:25)



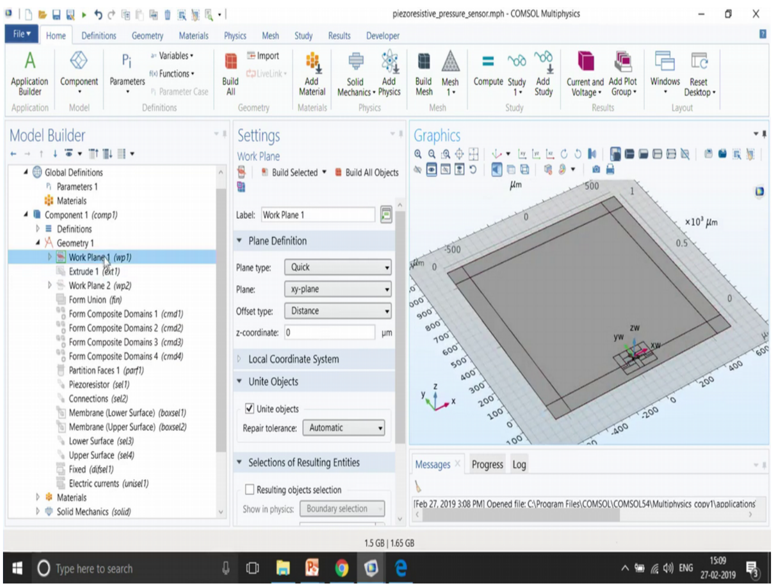
Again we use a tensor for both elasticity and coupling matrixes. And, so, let us go for the modeling part before we go to the results.

(Refer Slide Time: 24:39)



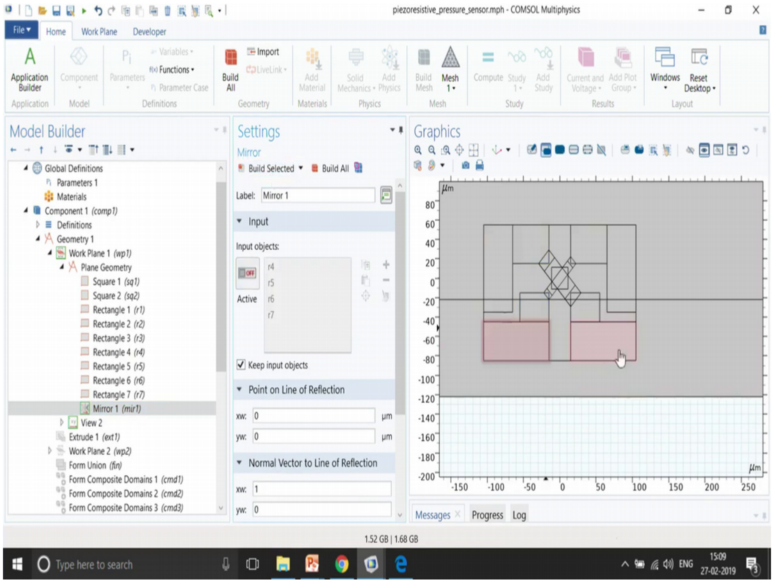
So, again the first thing is to make the geometry. So, here you will see now little bit complexity in the making of the geometry. Again, this is not that complex we have used some other domains to simplify the application of this particular domain so, while doing a physics. So, that is fine.

(Refer Slide Time: 25:01)



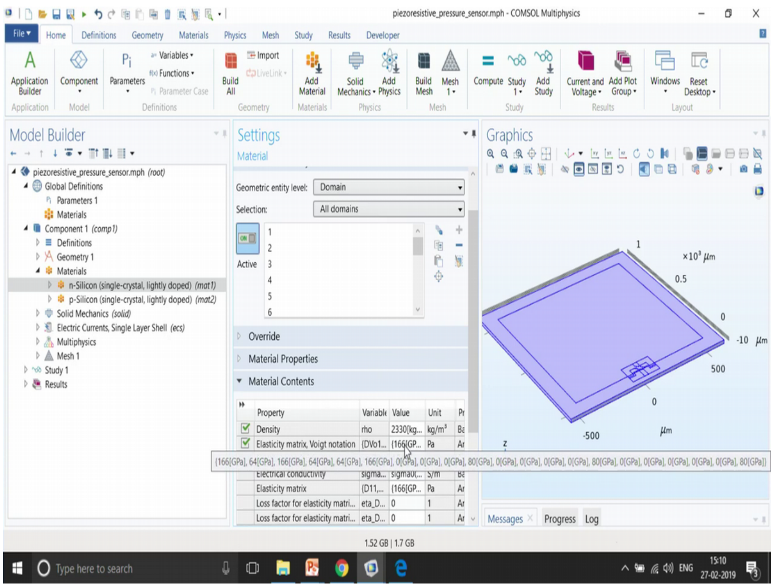
So, first thing is to make work plane and then if as you can see over here in the workplace we created a small-small geometries.

(Refer Slide Time: 25:08)



This is the first square that we made one more square that we made over here and then slowly and steadily we make the whole structure. If I just zoom in to this part so, slowly and steadily the complete structure was made like this right.

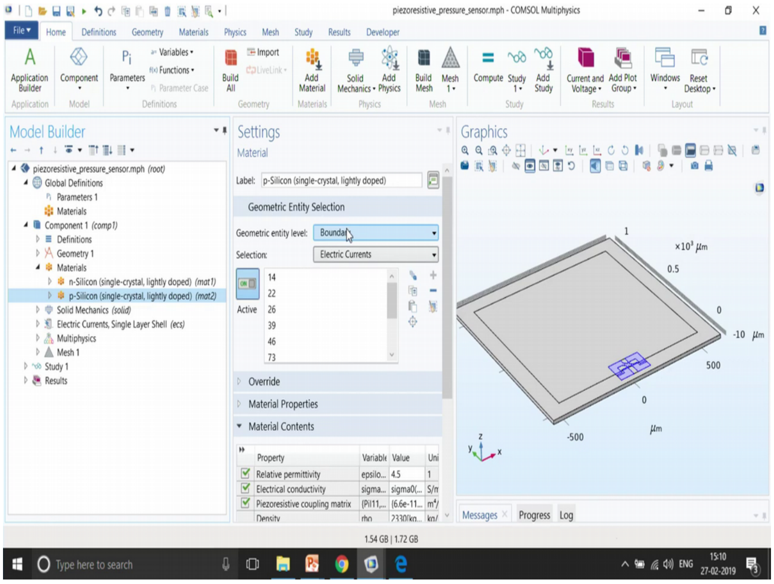
(Refer Slide Time: 25:29)



And, then we extrude this particular structure from top to bottom and then we do one more work plane I think this to extrude some ok. This is to extrude in the other direction. So once you are done with a geometry the most important part is to assign the materials.

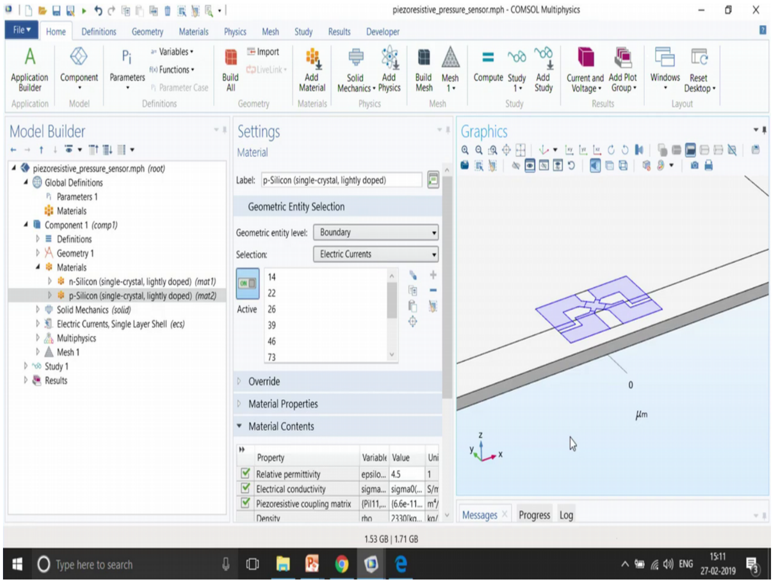
So, in this case you can see that we are assigning n-type silicon which is lightly doped to the complete domain. So, it is complete domain. So, we again talking about domains. It is very important we what kind of entity label that you assigning that particular domain right. So, right now the geometry entity level is domain that is volumetric domains in the 3D structure. In this case we have assigning scalar value of density and tensor for the elasticity matrix.

(Refer Slide Time: 26:25)



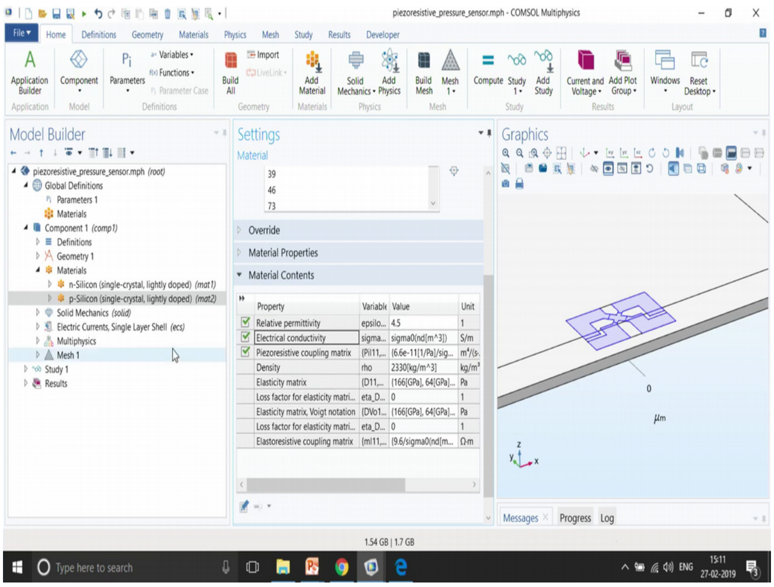
Again, for the p-type; now, the interesting part comes in to picture that we are using boundary. So, we have done away with the modelling volumetric approaches, but now we are going to model the p-type silicon as boundary.

(Refer Slide Time: 26:41)



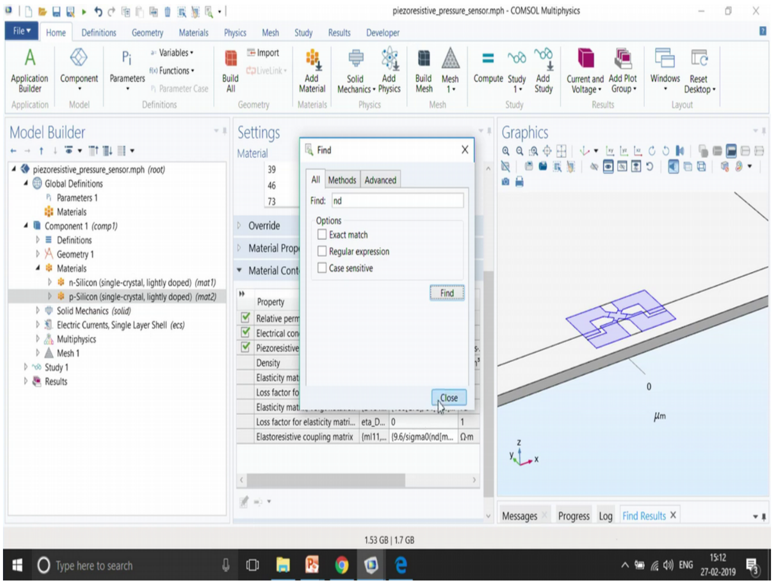
This very important and the reason why we are doing it is a machine, one of the main reason is the machine. You do not want us to spend a lot of time in meshing the domain if you are getting the similar kind of results in a very quick manner right. So, we use the geometry entity levels as boundaries not as domain, we are using as boundaries and selected this particular boundary.

(Refer Slide Time: 27:03)



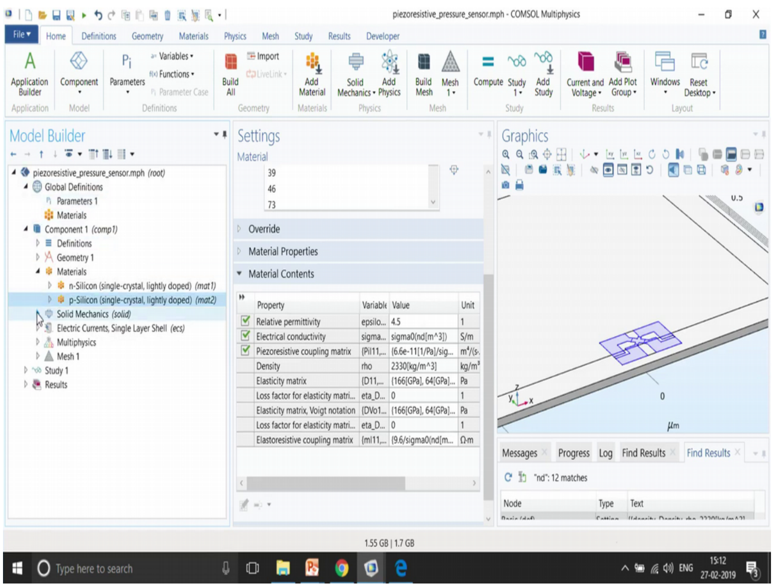
And, then we assign again the connectivity is again very interesting that the connectivity is a function of nd. So, we write sigma naught as a function of nd and this nd is the doping. So, from where the nd is defined? So, let me just go through it would be defined somewhere else.

(Refer Slide Time: 27:35)



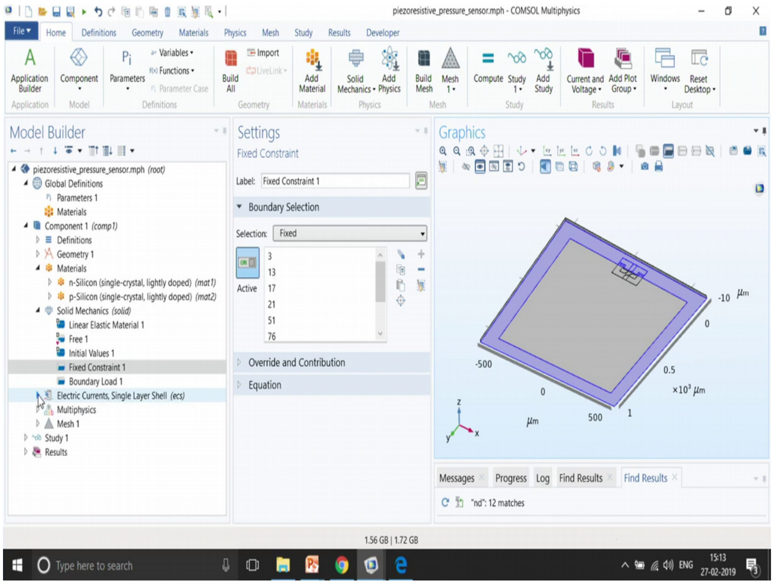
So, if you not find something this right do control F and then search for nd.

(Refer Slide Time: 27:55)



So nd is actually the density of the material, that is what understand. So, once you understand how the materials have been defined, so, over here also you have coupling matrix which is a 6 by 6 tensor.

(Refer Slide Time: 28:20)

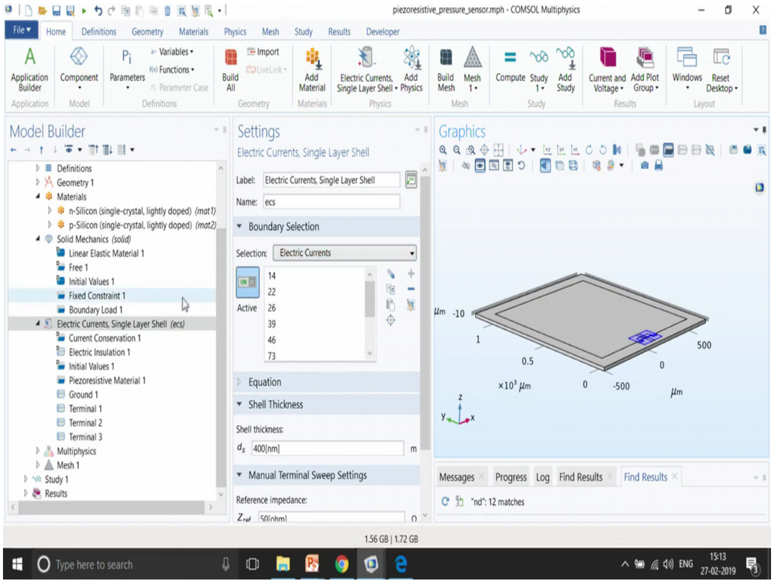


And, then the next part is the physics. So, in this case you can see that solid mechanics domain has been assigned. In the case of shell only a shell would be assigned. We give a fix constants on the bottom part; this is a bottom red part that you can see over here. So, in the boundaries we give fix constant and give a boundary load on the top part.

So, a force has been applied in the inner surface over here and what force? We are giving 100 kilo Pascal of force that is very important of how much force we are going to give. And, we are giving a fix constant in the bottom outer edges.

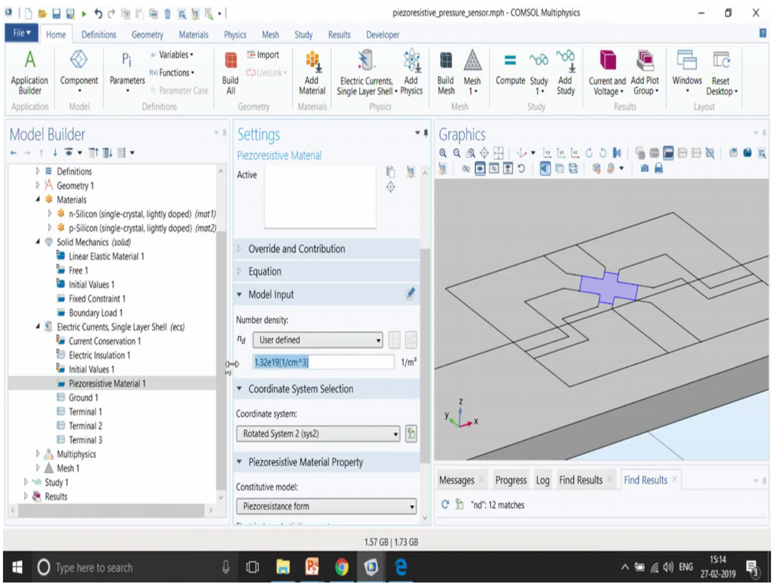
If you have any questions please write it down in the question answer session and we will try to respond you through the forum.

(Refer Slide Time: 29:13)



The next part the most important part is now how do we handle the electric currents in a boundary? So, again you will see that we are done away with the volumetric approach.

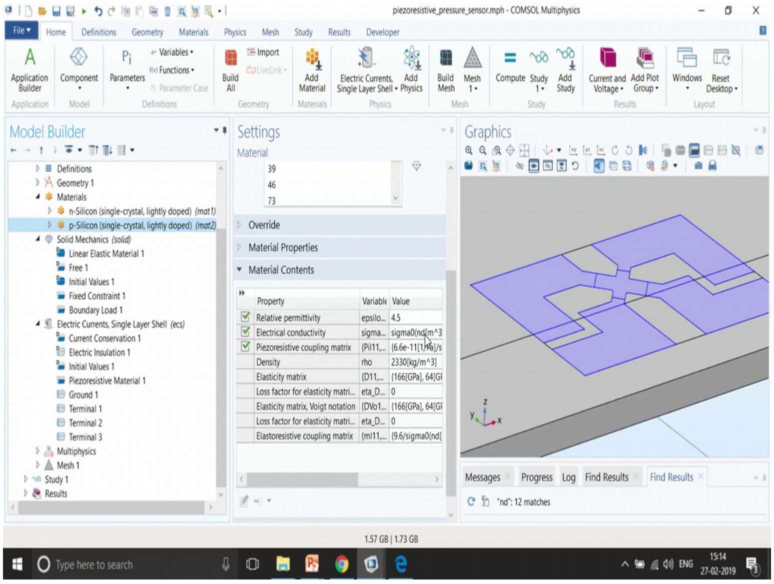
(Refer Slide Time: 29:22)



And we are only if I just zoom in we only talking about boundaries. These are all boundaries so, that is why the reason shell has been mentioned over here right and we also mention the thickness of the shell. So, this is in the kind of approximation. Why do you want to go for a volume which is having 400 nanometers? This is going to take a lot of time from meshing.

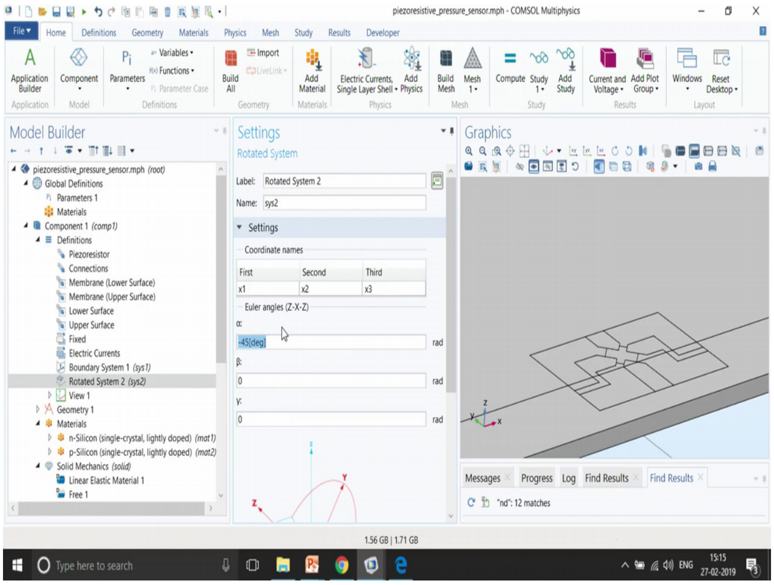
So, and forget about solving so, solving will also take more time. But, if you have an approach where you can minimize this meshing issues then it is always good. So, that is why we use shell layered approach. In this case again we use Piezoresistive Materials that you can see over here. So, this is a Piezoresistive Material small cross shaped model over here and over here is the number density which is defined that I was searching before alright.

(Refer Slide Time: 30:18)



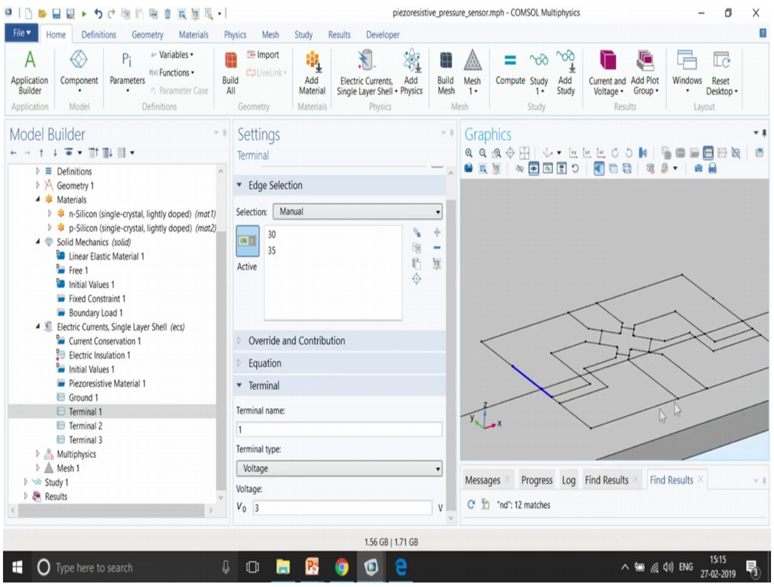
So, the number density is defined over here. And, the conductivity the conductivity is actually a function of nd the number density all right. And, then again it is using a rotated system. So, again is not using the default coordinate system, but it is using the rotated systems. So, it is very important to know that a rotated a rotated system has been defined over here.

(Refer Slide Time: 30:46)



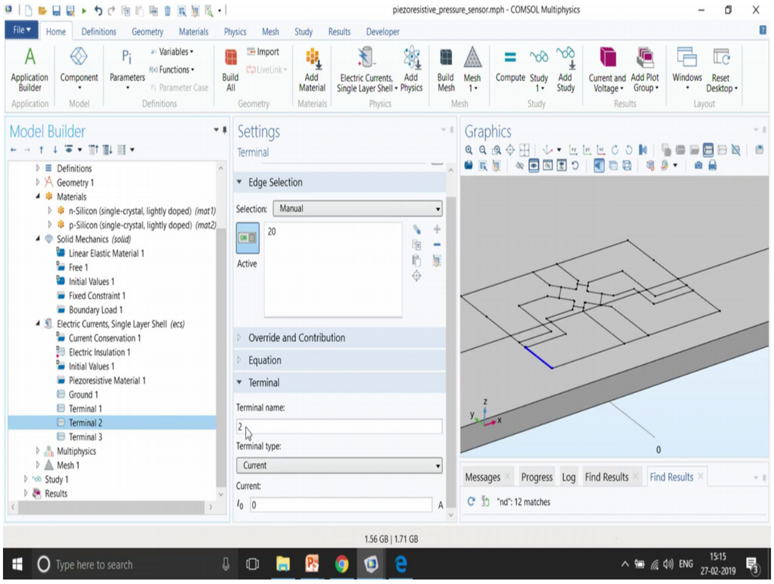
So, what kind of angle cut has been given in this particular case? So, minus 45 degree cut has been given over here.

(Refer Slide Time: 30:56)



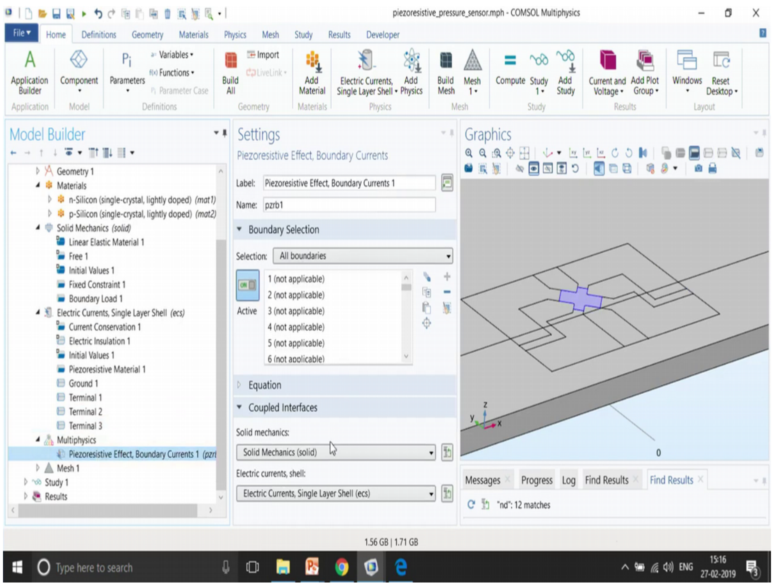
So, in the next part is from where the potential has been given. So, you can see in the terminal 1 a particular voltage of 3 volts have been applied over here. So, I am zooming a part of piezoresistive materials only. So, terminal boundary condition have been given over here and a ground has been given on the opposite side. So, the current is going to flow like this to this domain to the piezoresistor and then finally, it will arrive over here.

(Refer Slide Time: 31:27)



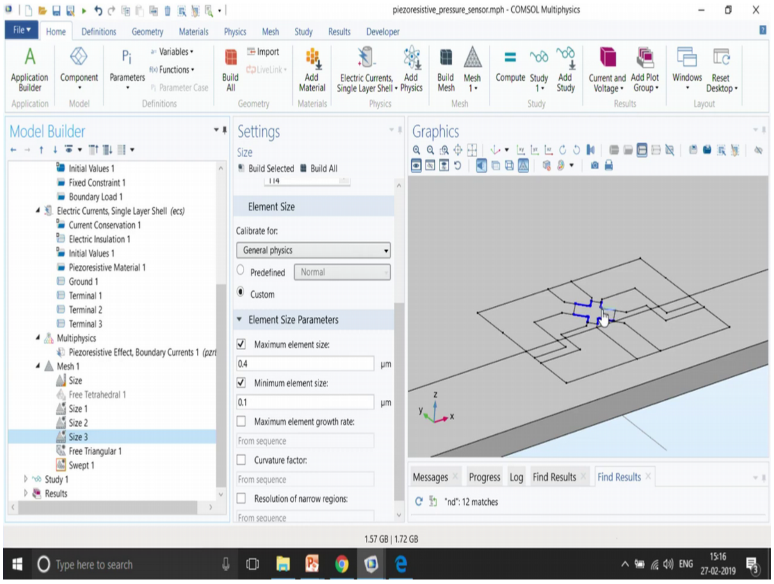
The next terminal 2 has been given zero current and terminal 3 is also given zero current. This is to reduce the error while we are taking the measurements ok.

(Refer Slide Time: 31:42)



So, next part is again the multi physics part that couples the structural mechanics with the electro electric currents interface.

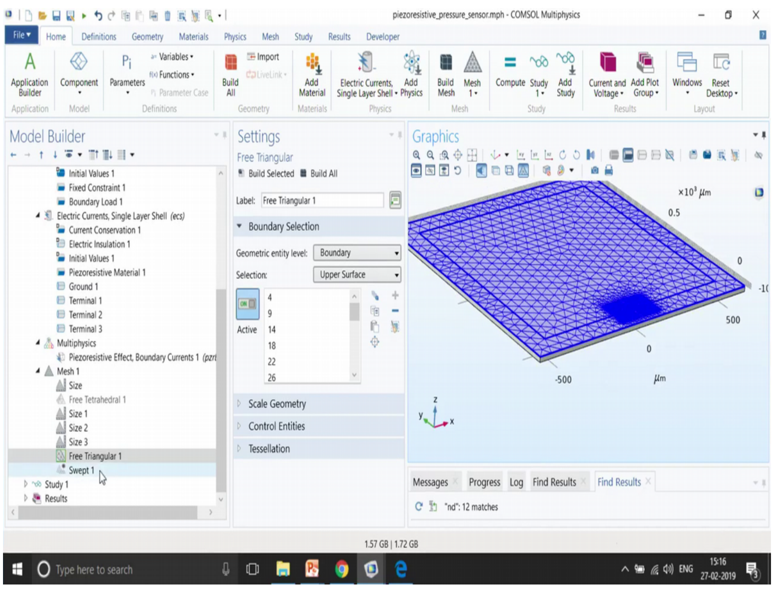
(Refer Slide Time: 31:47)



And, then finally the mesh part this mesh part is always interesting because there are many different ways or approaches to mesh this particular domain. The way it has been model over here is they have given different different meshing elements to each of the domains. So, you can see the piezoresistive material is having maximum elements size is 2 micro meters.

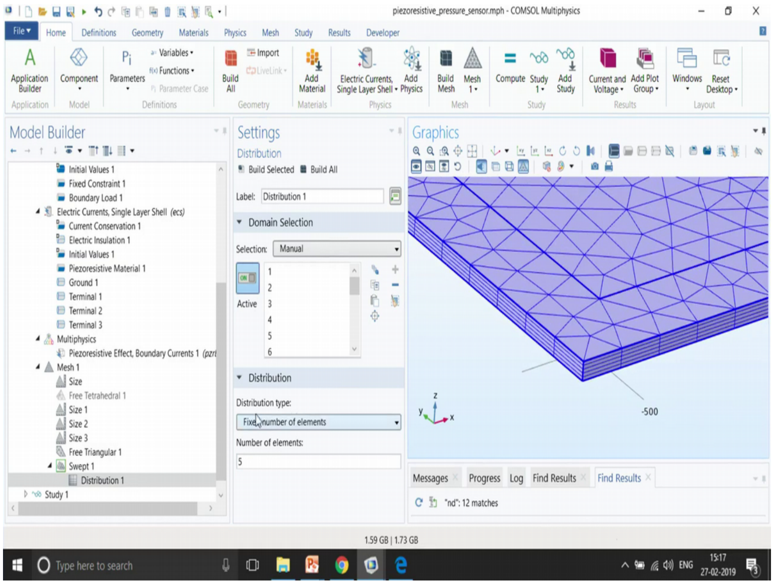
The other elements of the piezoresistive material has been given with 6 micro meters and then the edges also have been model over here with the different elements size.

(Refer Slide Time: 32:22)



Andm, then the free triangular top boundary has been model. So, not the complete volume, but only the top boundary has been modeled over here.

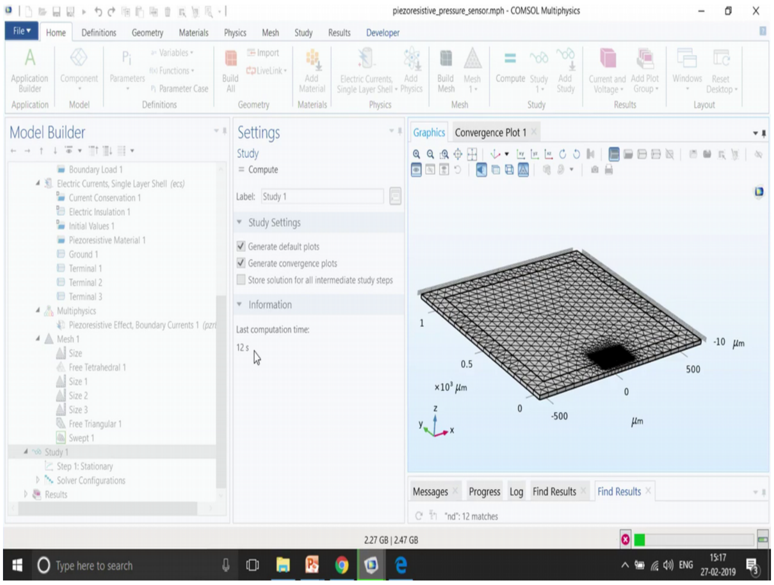
(Refer Slide Time: 32:36)



And, then you are sweeping it from top to bottom right. So, right now this is only one sweep. This is one domain with which it is sweep. If you want to give it (Refer Time: 32:42) of five domains that you want to see from the top to bottom sweep. So, you can just write it as number of elements as 5. And, then we go to study and then run the simulation alright.

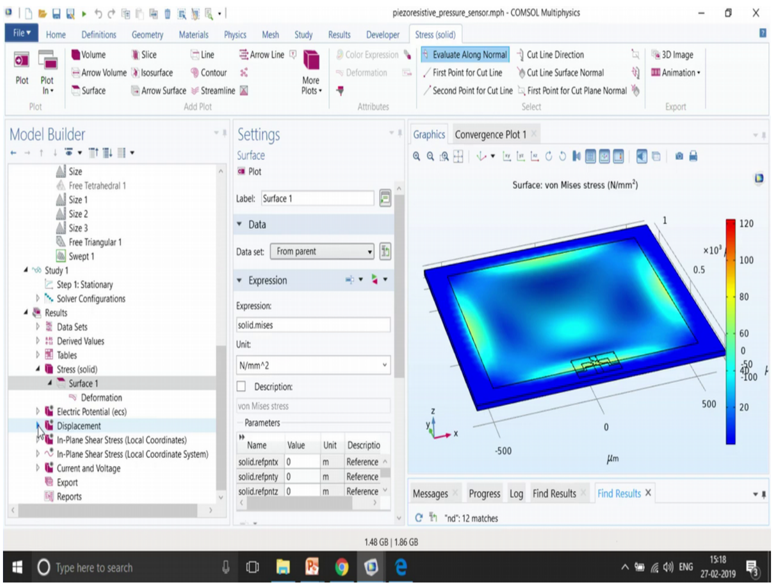
So, let me just delete it and keep whatever is by default so that stimulation results could be performed very quickly.

(Refer Slide Time: 33:03)



So, as it says that around 12 seconds to it will take, so, it is not that much of time all right. So, I can just go ahead and run the simulation.

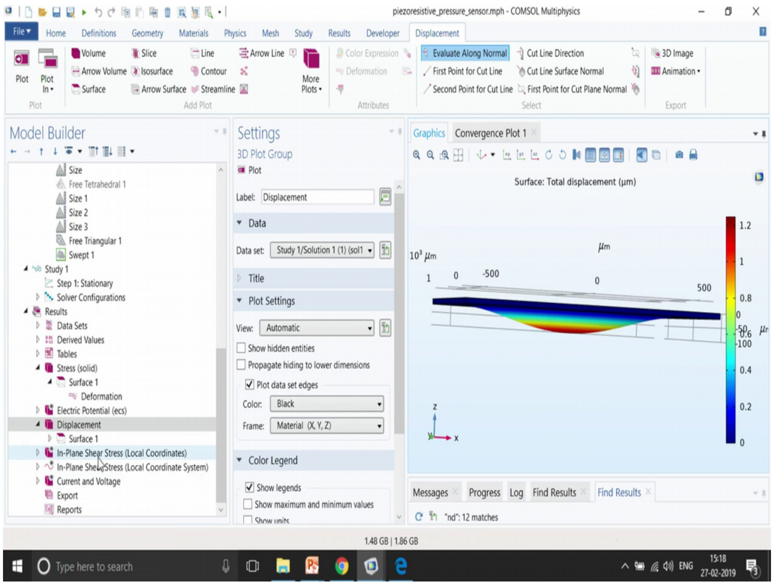
(Refer Slide Time: 33:14)



So, now you can see the stresses that is being developed on the surface and along with the deformation. So, says you how the deformation is going to be and this is a skilled version of deformation. So, if you want to see the actual deformation, make the Scale factor as 1. The deformation that the this stresses that you can see you can also change the unit from Newton per meter to Newton per mm square and then plot right. So, it is around maximum is around 120 Newton per mm square.

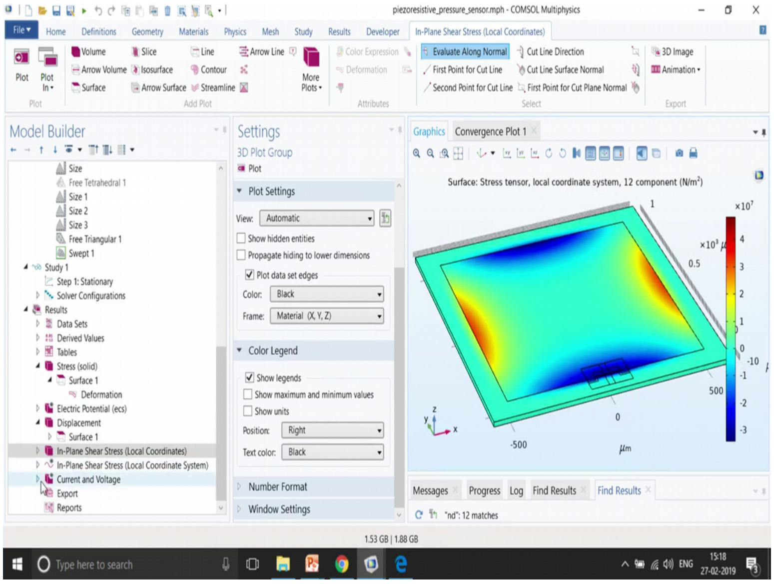
And, in addition to it you can also know the factor of safety with which your system would be working. So, your device should not be going more than the factor of safety. So, that is also kind of analysis that you can do, if your structure is going to fail or not.

(Refer Slide Time: 34:11)



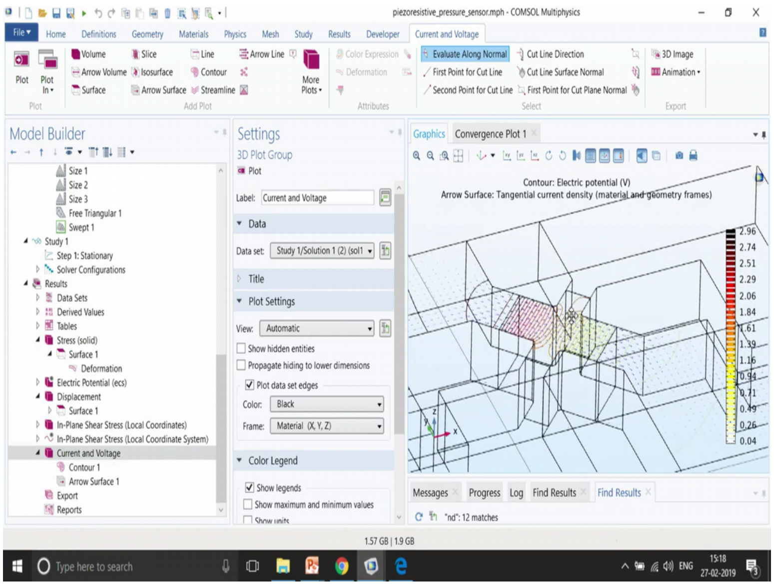
This plot talks about the displacement. This makes sense because it is having bounded surfaces on the edges. So, all the forces are going to accumulate in the center as you are going to pull it down on the center.

(Refer Slide Time: 34:24)



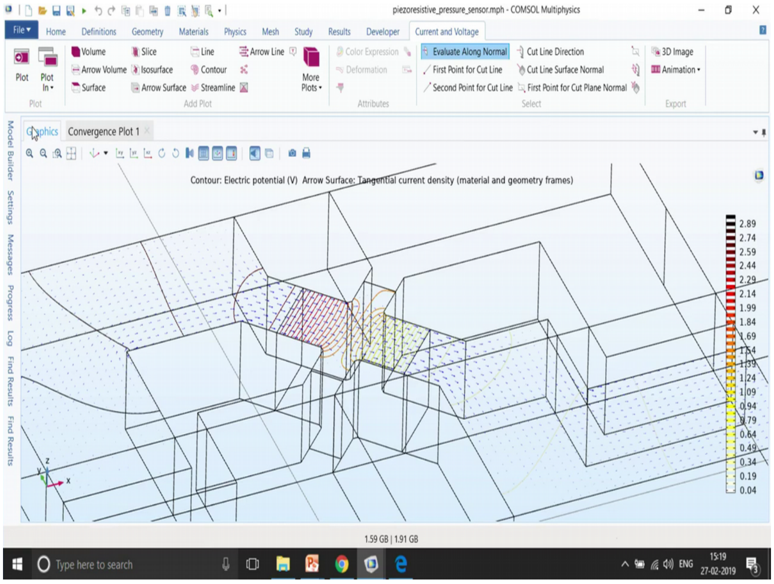
So, we have in plane stresses also that you can mod see over here.

(Refer Slide Time: 34:29)



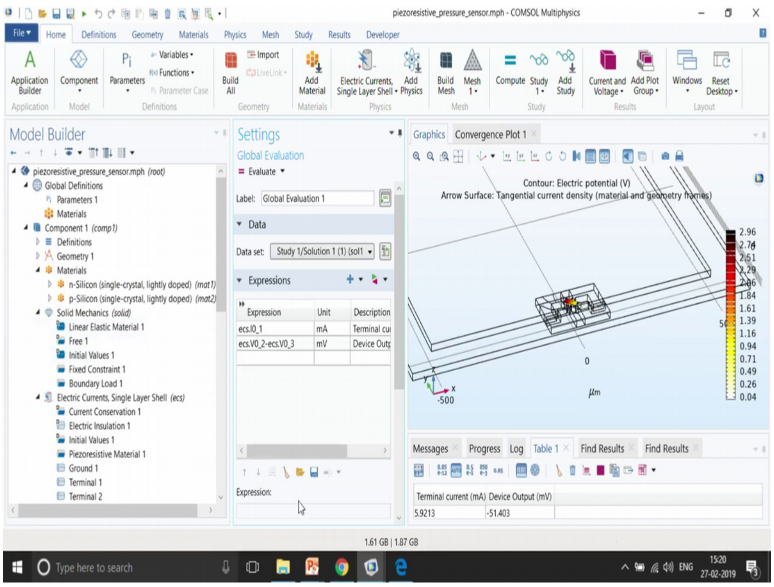
And, then finally, the currents and the voltages right so, this is how the currents are going to move. Please maximize it.

(Refer Slide Time: 34:34)



So, you can see that the voltage that I have applied is over here somewhere over here and the colour represents the per volt sorry the colour of these counter plots represent the voltage that you can see over here. And, the arrow plot actually represents the flow of the current from the voltage part over here to the ground over here. So, this is the flow of current that you can see right.

(Refer Slide Time: 35:12)

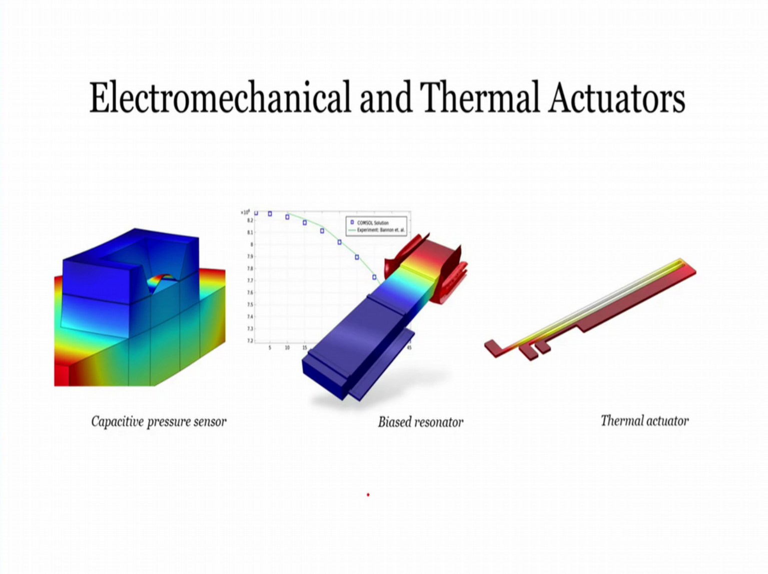


So, you can actually first make this particular design with COMSOL and then actually we make optimize your design to see which is the highest current that actually flows through this device and then finally, you can fabricate your own device. If you want to see quantitatively how much current is going to flow, you can do a global evaluation that you can see over here.

Just search for current over here. So, I just go to replace expression, current over here and you can see how much current for all the terminals that you can do it over here ok.

So, this is how the approach to model piezoresistive models are ok.

(Refer Slide Time: 36:04)



So, let us go to the next section is electromechanical and thermal actuators.