INDIAN INSTITUTE OF SCIENCE, BANGALORE

NPTEL

NATIONAL PROGRAMME ON TECHNOLOGY ENHANCED LEARNING

NPTEL IS OFFERING ONLINE CERTIFICATION COURSES

CERTIFICATES FROM THE
IITs &IISC
Are just a click away…

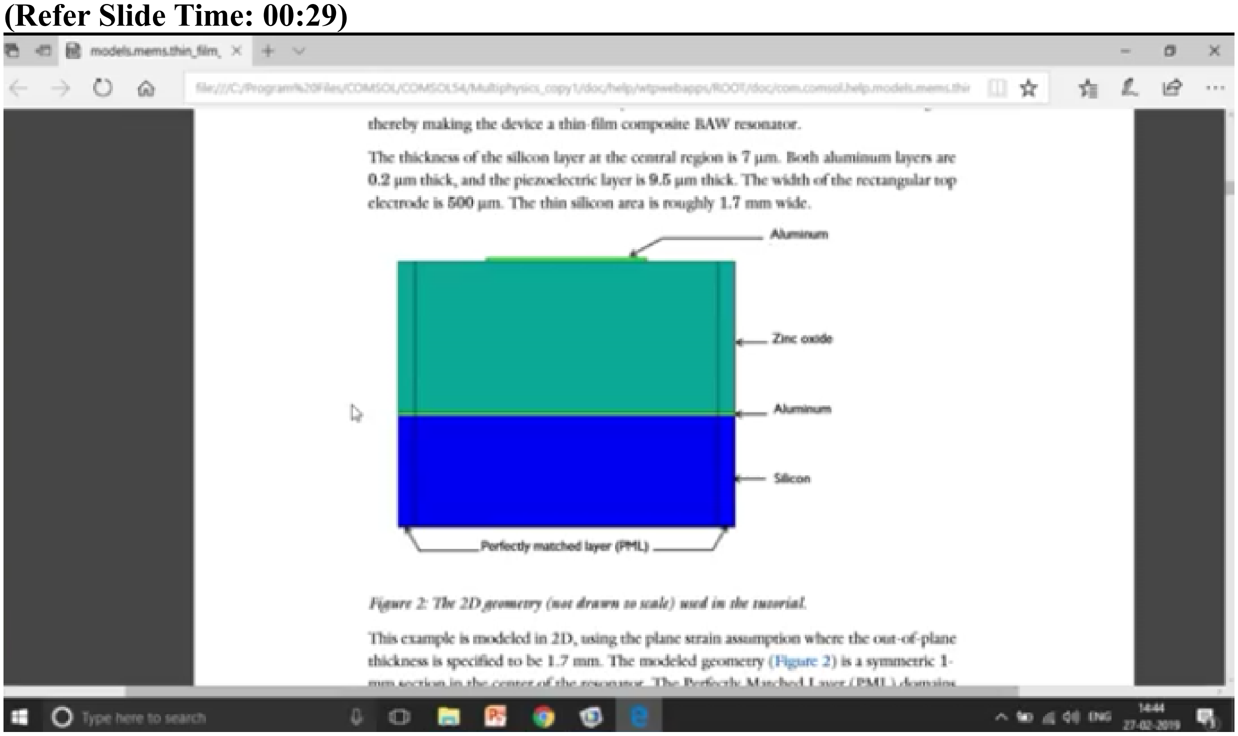
NO ENTRANCE EXAMS, NO ENTRY LEVEL CRITERIA
JOIN ANY COURSE. ENROLL FOR FREE!!

https://onlinecourses.nptel.ac.in/

Course Title
Electronic Modules for Industrial
Applications using Op-Amps

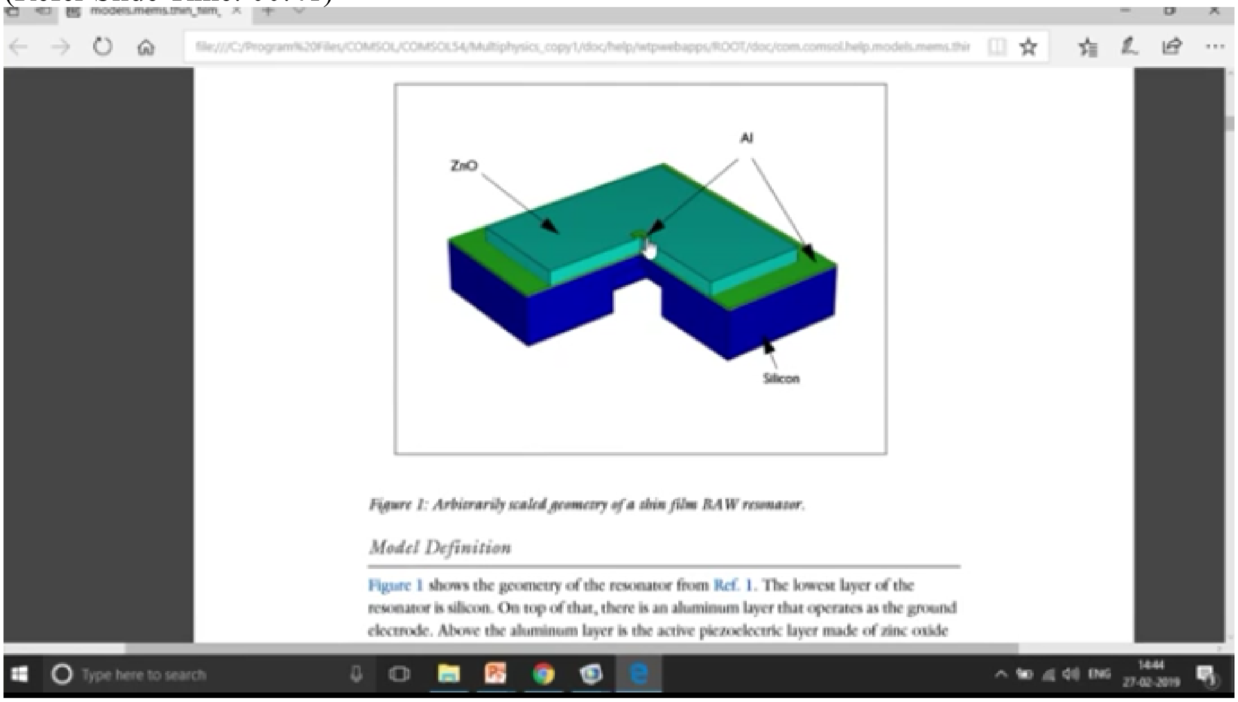
By
Dr. Hardik J. Pandya
Department of Electronic Systems Engineering

(Refer Slide Time: 00:29)



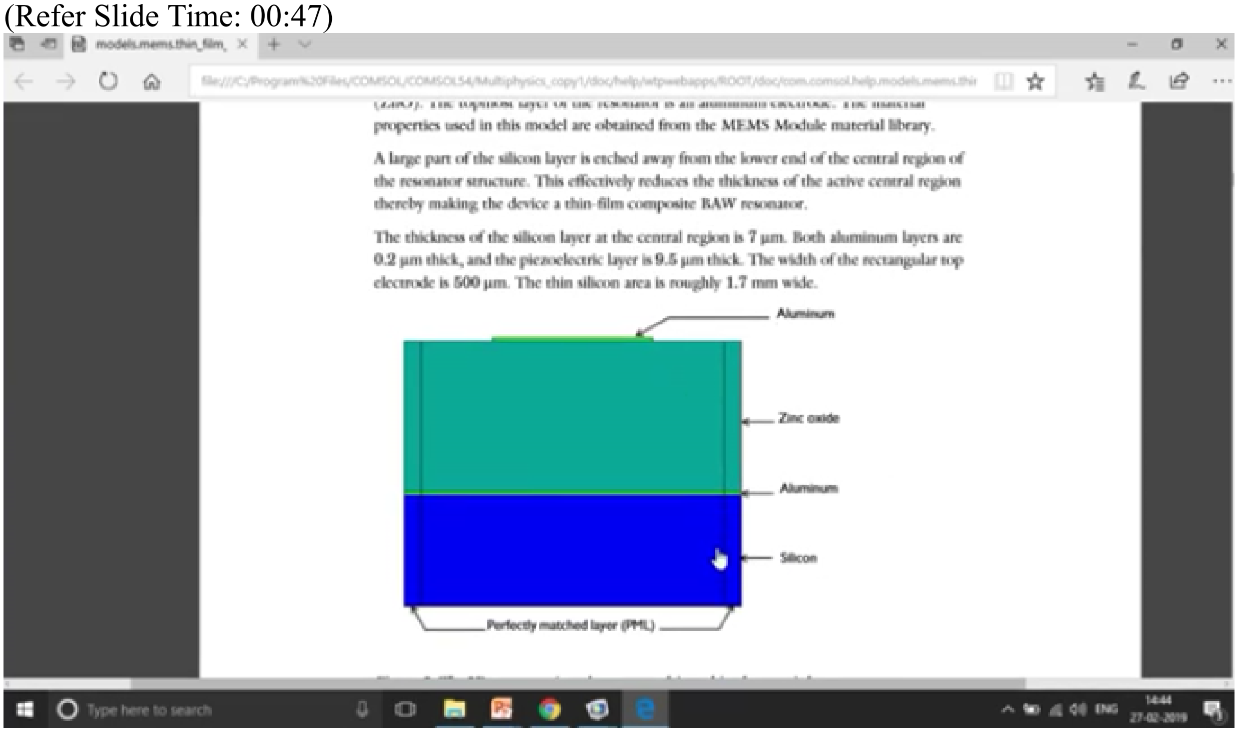
Okay, and in addition to it as we saw in the earlier example periodic structure where model held, but in this case there is no case of periodicity,

(Refer Slide Time: 00:41)



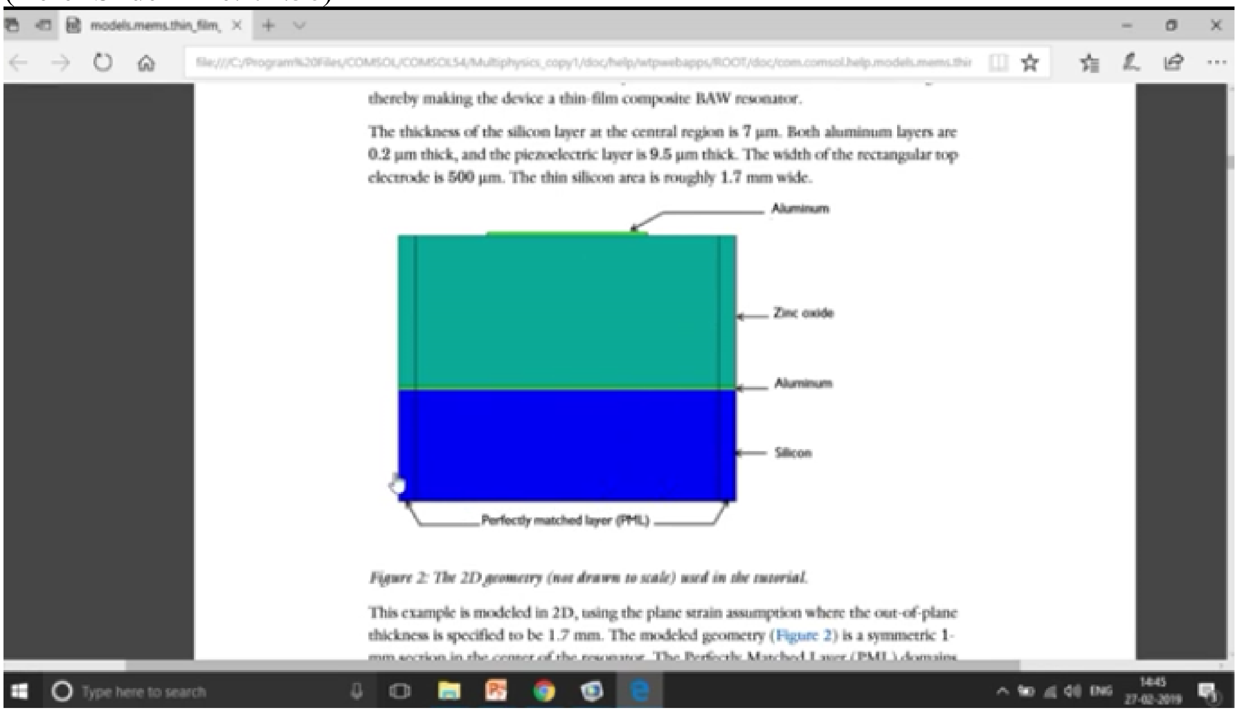
it's not above structure of aluminum is not actually getting repeated.

(Refer Slide Time: 00:47)



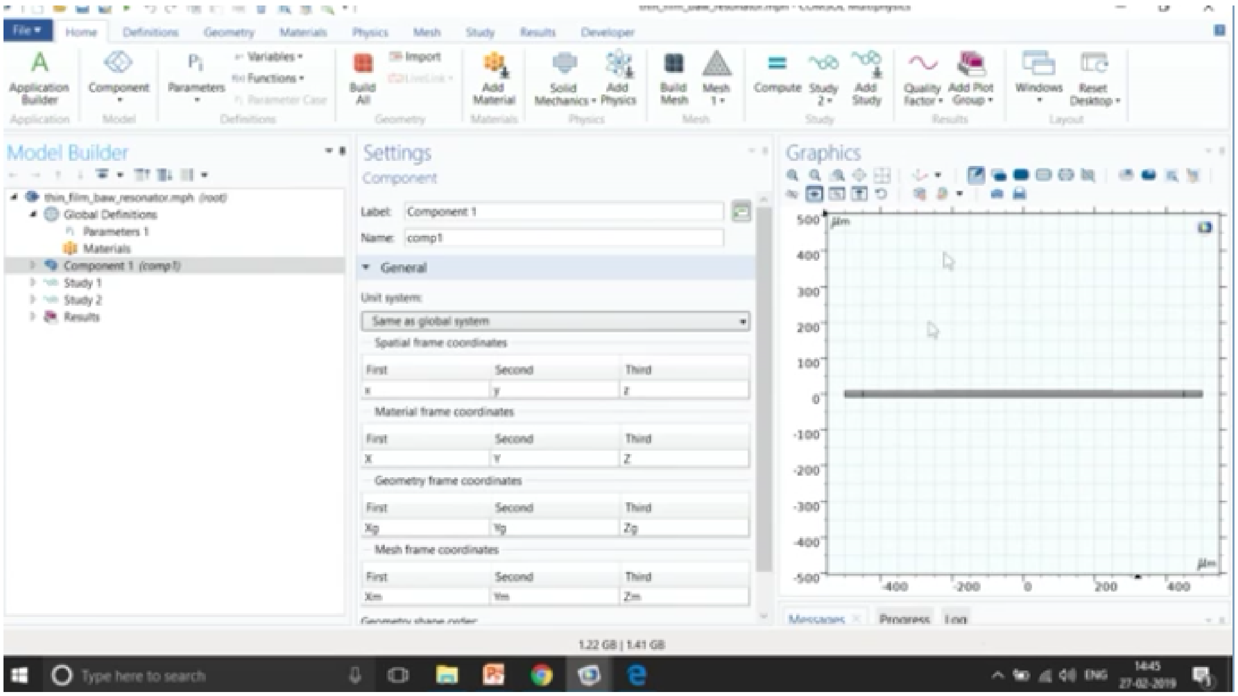
In this case hence we have used PML's on the left and right, so the forces that are going to go for the electric currents that are going to flow, I'm not going to get reflected back from this boundaries, we should not forget that this whatever we are doing is a mathematical approach of actually solving 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 you 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 we will talk about in some time.

So this is the actual structure aluminum zinc oxide, aluminum silicon,
(Refer Slide Time: 01:38)



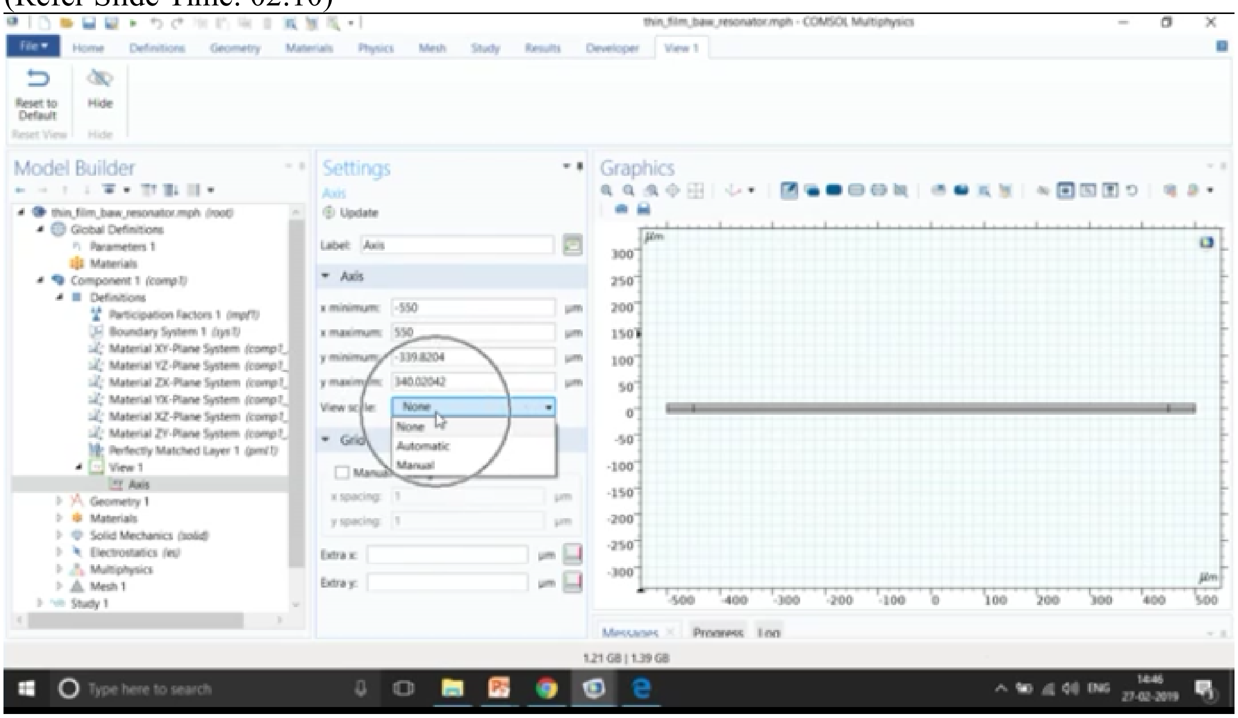
the same thing in 2D as we model as aluminum, zinc oxide, aluminum, silicon, and finally PML on the left and the right side.

(Refer Slide Time: 01:50)



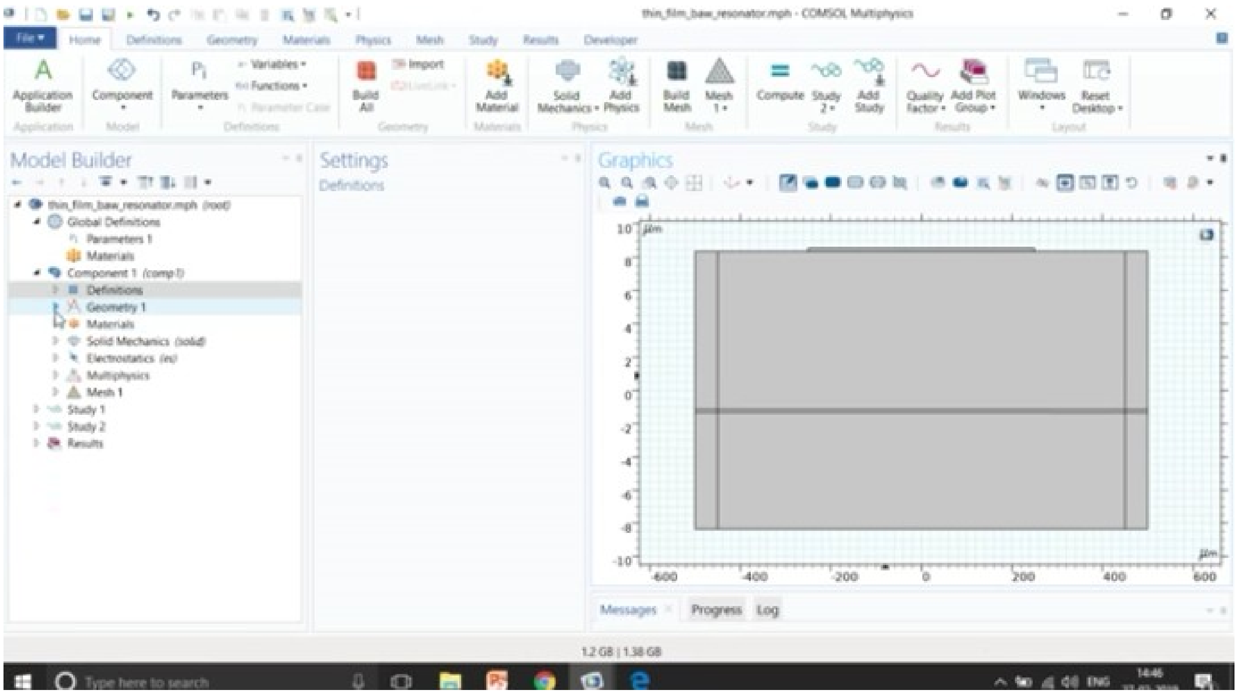
So now let me go and open the model, so right now you can see that your material, this is the actual geometry, so in COMSOL you can also improve the view of your geometry by going to this definitions, view, in the view you can go in to axis and view scale as automatic,

(Refer Slide Time: 02:10)



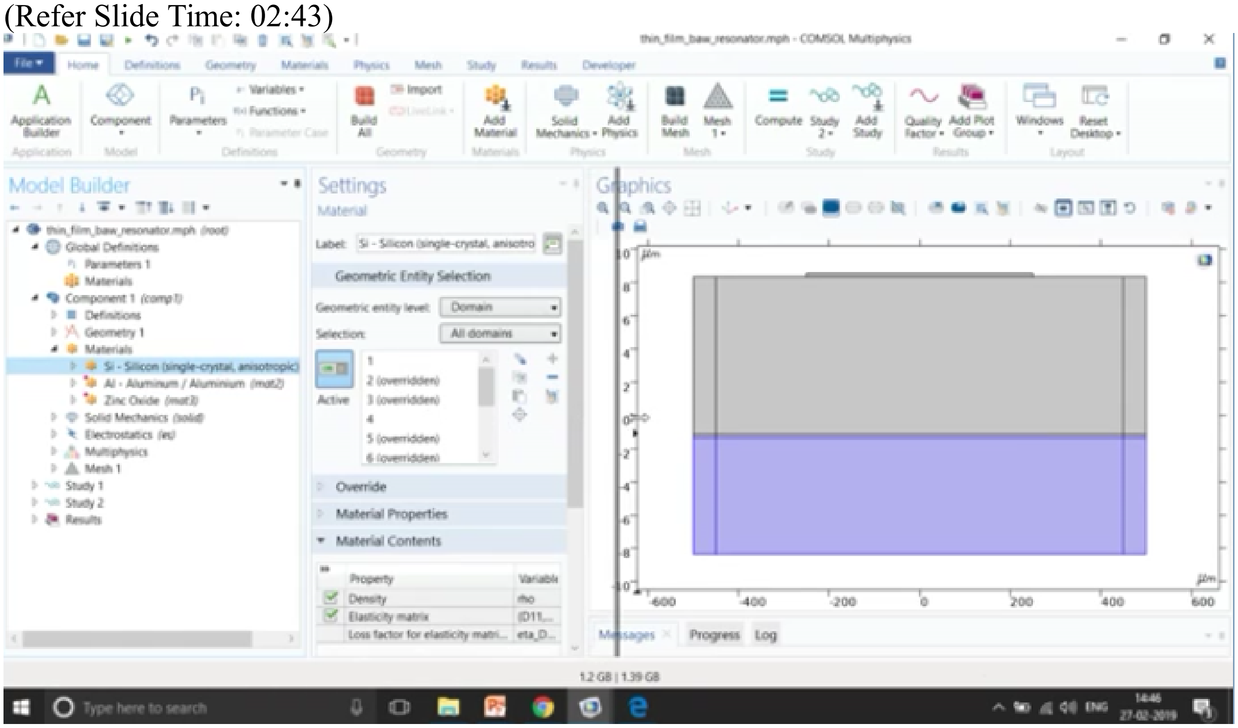
this will actually scale your geometry to little bit large, so your actual geometry size remains the same, but the view of your geometry actually improves, so that you can select those particular domains, you try to understand what physics you want to apply.

(Refer Slide Time: 02:29)



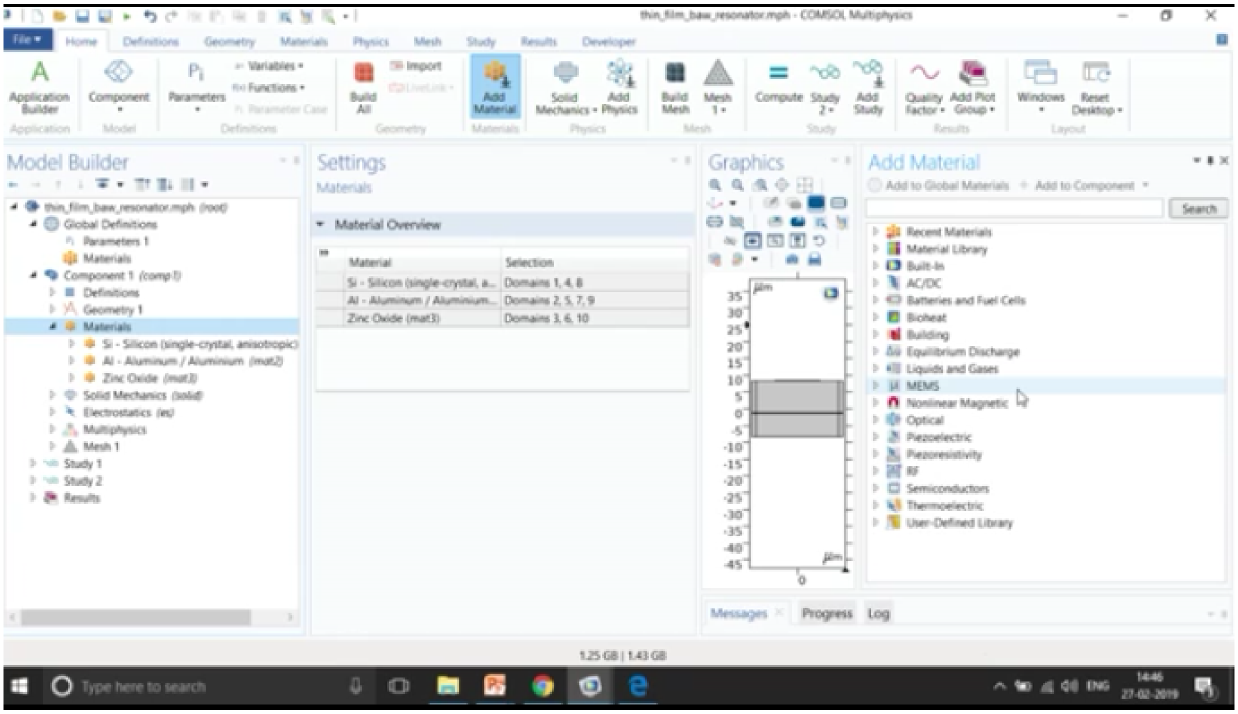
So the first thing is to make the geometry, so I use rectangle 1, rectangle 2, rectangle 3 to make the geometry, the next part is the materials, the most important part is the materials, here the silicon material has been modeled,

(Refer Slide Time: 02:43)



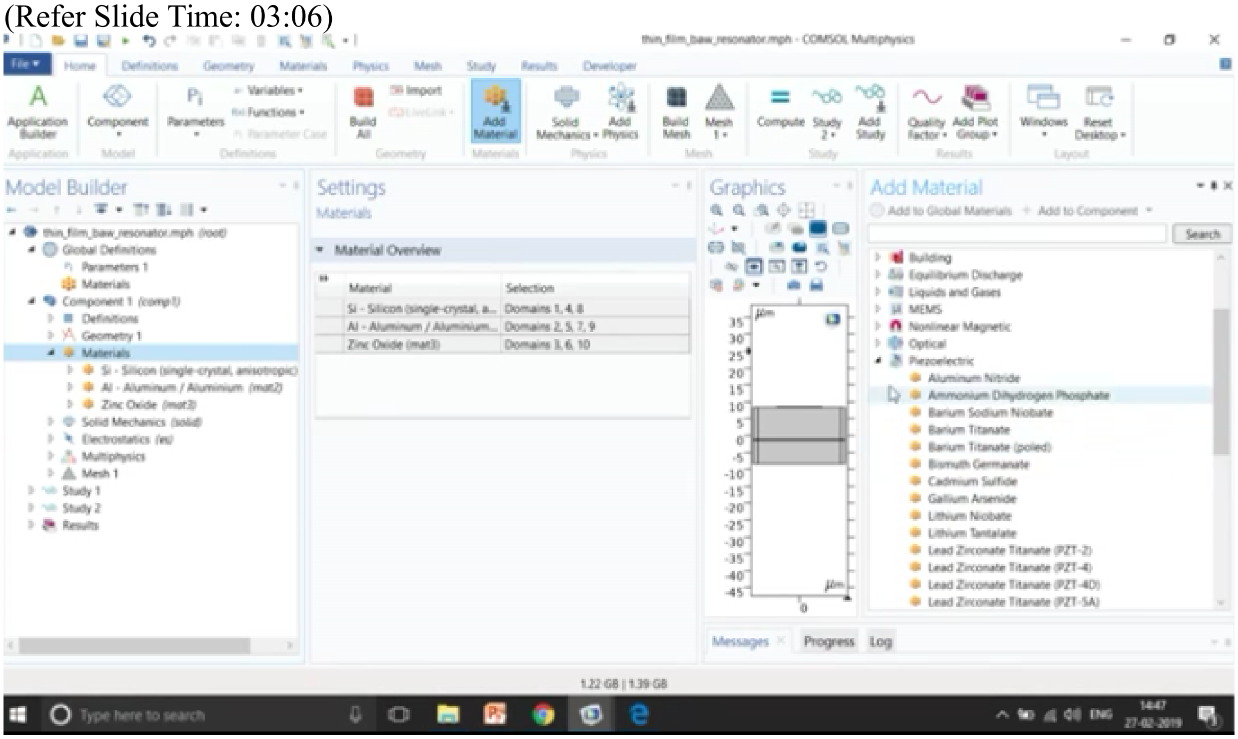
so we already have many different materials, so if you just right click on material, add material from library, and you will see that there are many different materials, and most of the times you would get the materials that you require from this material library,

(Refer Slide Time: 03:03)



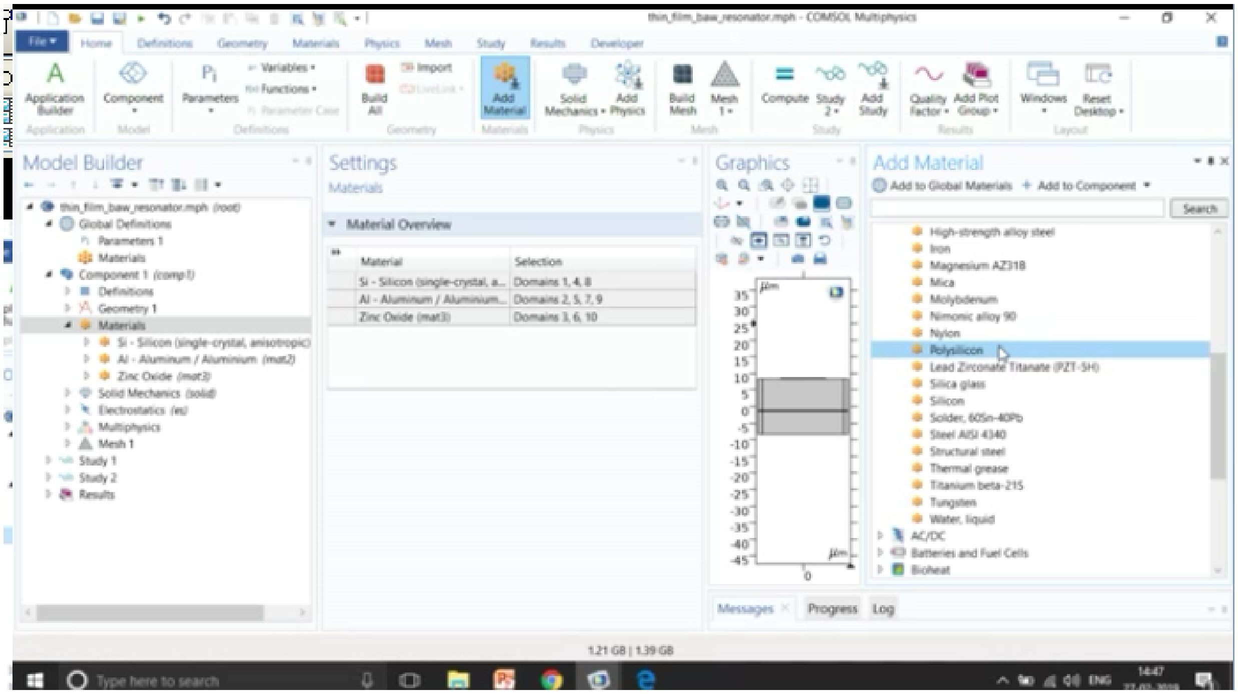
so you have in piezoelectric, there are many different types of piezoelectric material available as you can see over here.

(Refer Slide Time: 03:06)



Silicon is available in the building, so you will have a silicon over here somewhere, yes silicon over here, in addition to it we also going to do a domain thermal actuator,

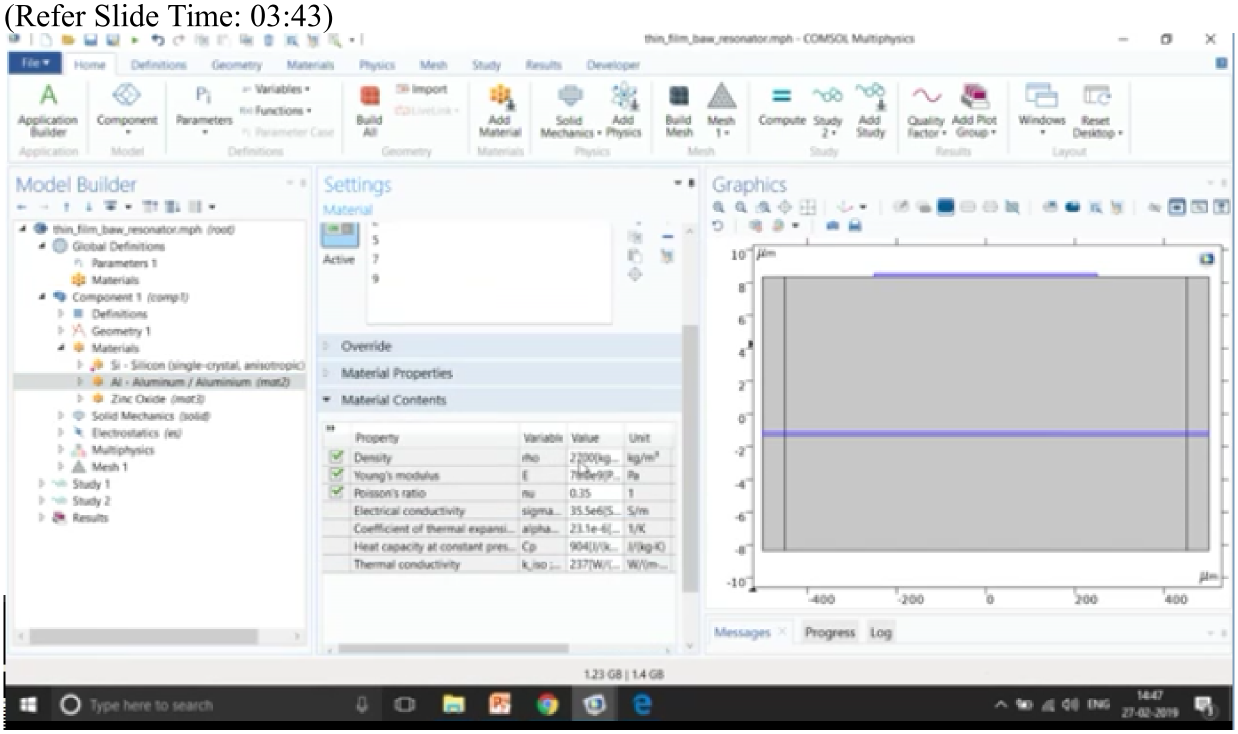
(Refer Slide Time: 03:21)



and polysilicon is going to be used in the thermal actuator.

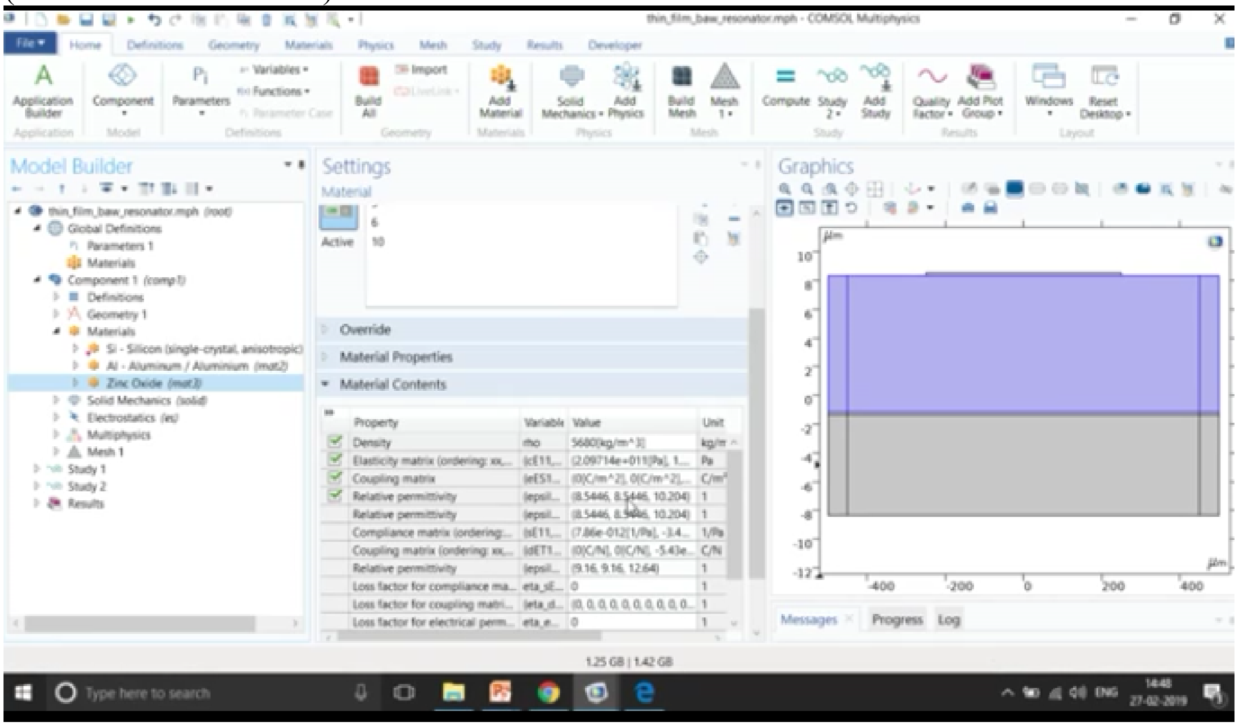
So let's go to silicon that is the bottom part, and you can see the density in elasticity matrix with which the silicon has been modeled, again a tensor with which the elasticity mass matrix has been defined for silicon, then the thin aluminum metal contacts,

(Refer Slide Time: 03:43)



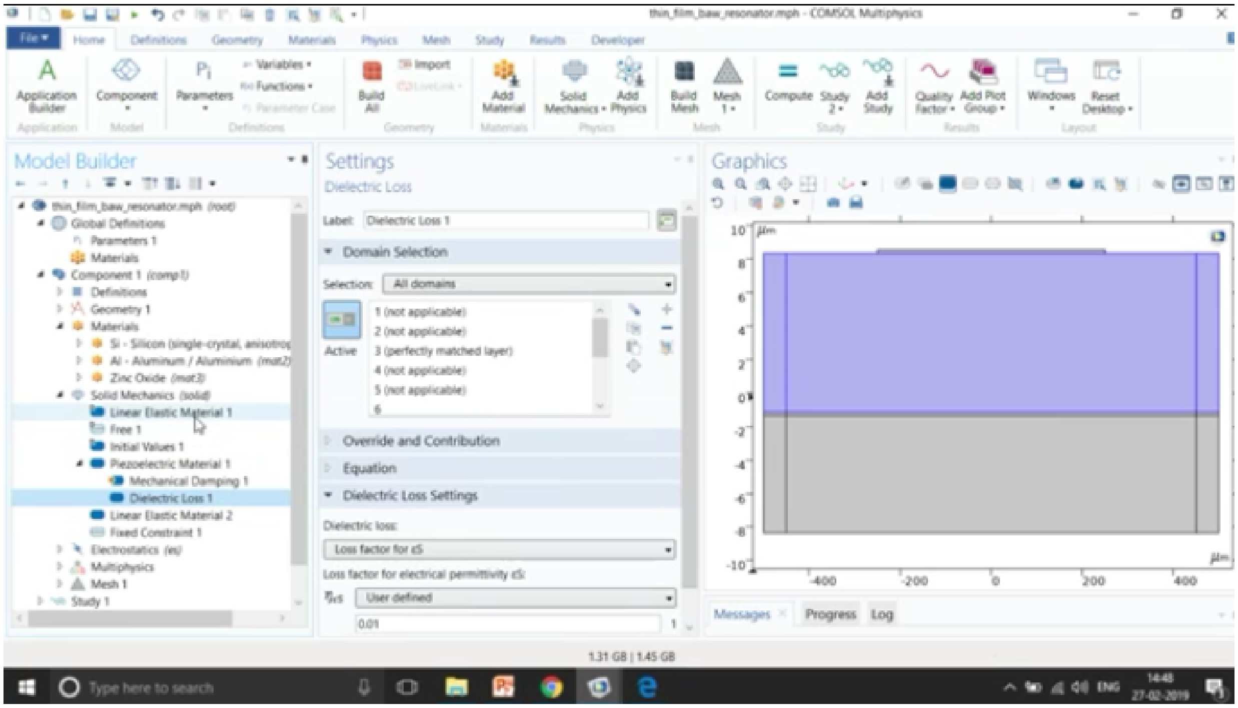
this one you could see over here, this is again the scalar material properties, and then the zinc oxide material that you can see over here, this is again elasticity matrix, coupling matrix is a tensor, and again the relative permittivity is a anisotropic which is more in Z direction as compared to X and Y.

(Refer Slide Time: 04:09)



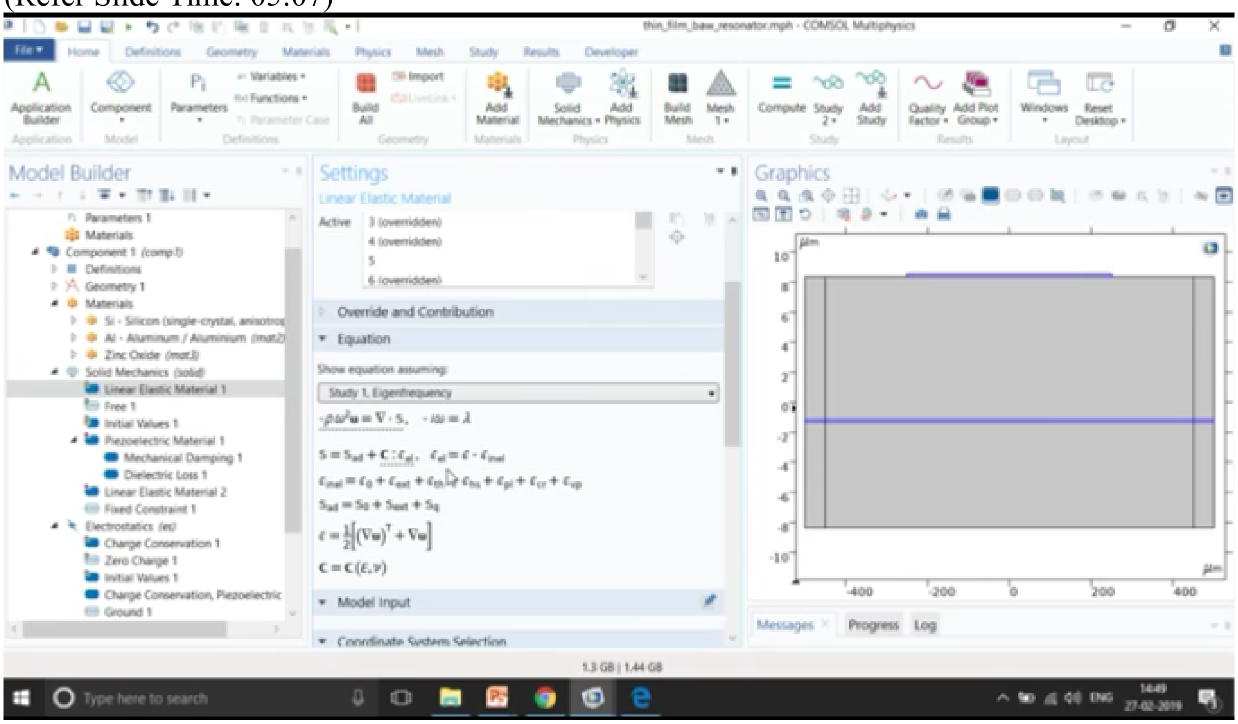
The physic set up is should be simple, so we are using for zinc oxide you can see over here, we are using piezoelectric material property over here and then we are using stress charge form, and in addition to it there are some kind of losses, there could be mechanical damping or some kind of dielectric losses, because which is actually hampering the forces that is going to happen, so such kind of losses could also be taken into consideration.

(Refer Slide Time: 04:42)



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 the Poisson’s equation, so it actually shows the equation, for example in 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,

(Refer Slide Time: 05:07)

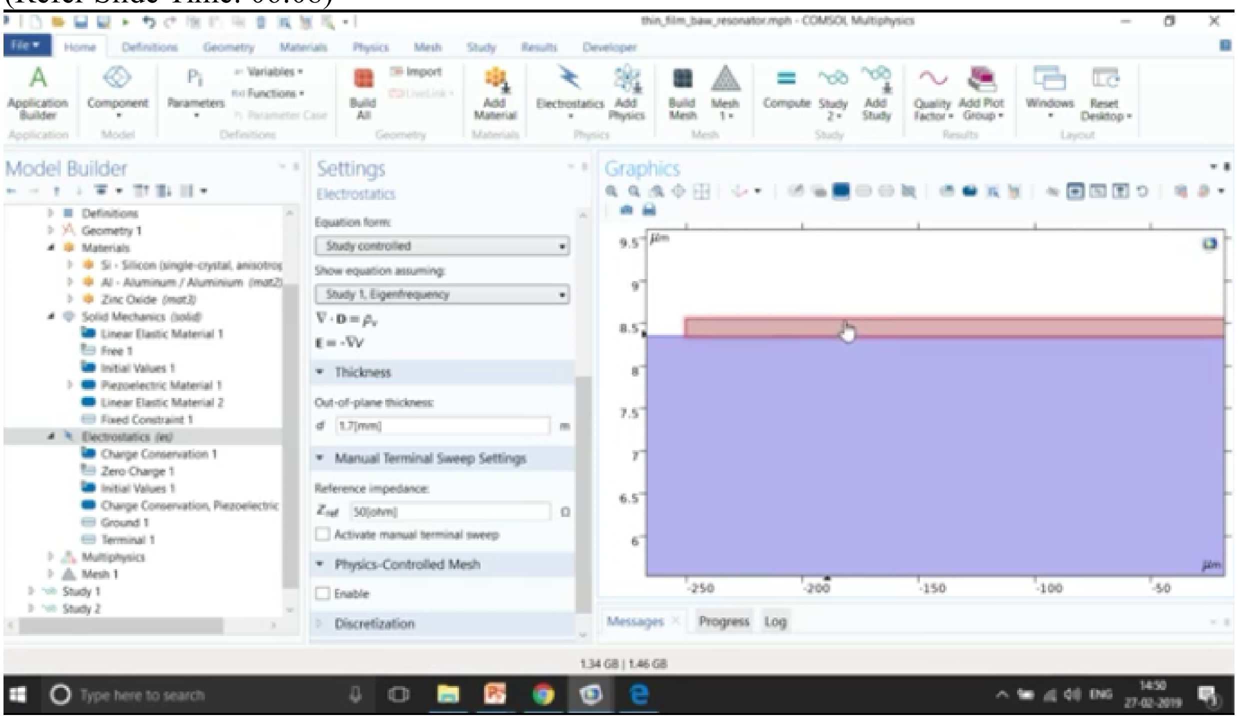


it doesn’t hide you what is it solving for, in piezoelectric material also you will see all the equations are available over here, and the dotted part that you can see over here is actually what you are giving through this particular setting mode, from this particular setting mode, the dotted part is what you are giving from the this particular setting mode.

And along with the piezoelectric material, we are modeling the linear elastic material for silicon bottom which is an anisotropic material. These are the different types of material models which are available.

And then finally we are doing fixed constants on the left and right side, then the electrostatics part where we are only solving for the top piezoelectric material, and we are again giving a piezoelectric material model for the top part, we are giving a terminal boundary condition, so here it's important to know that we have not selected the aluminum part within the electrostatics, so this is the aluminum part,

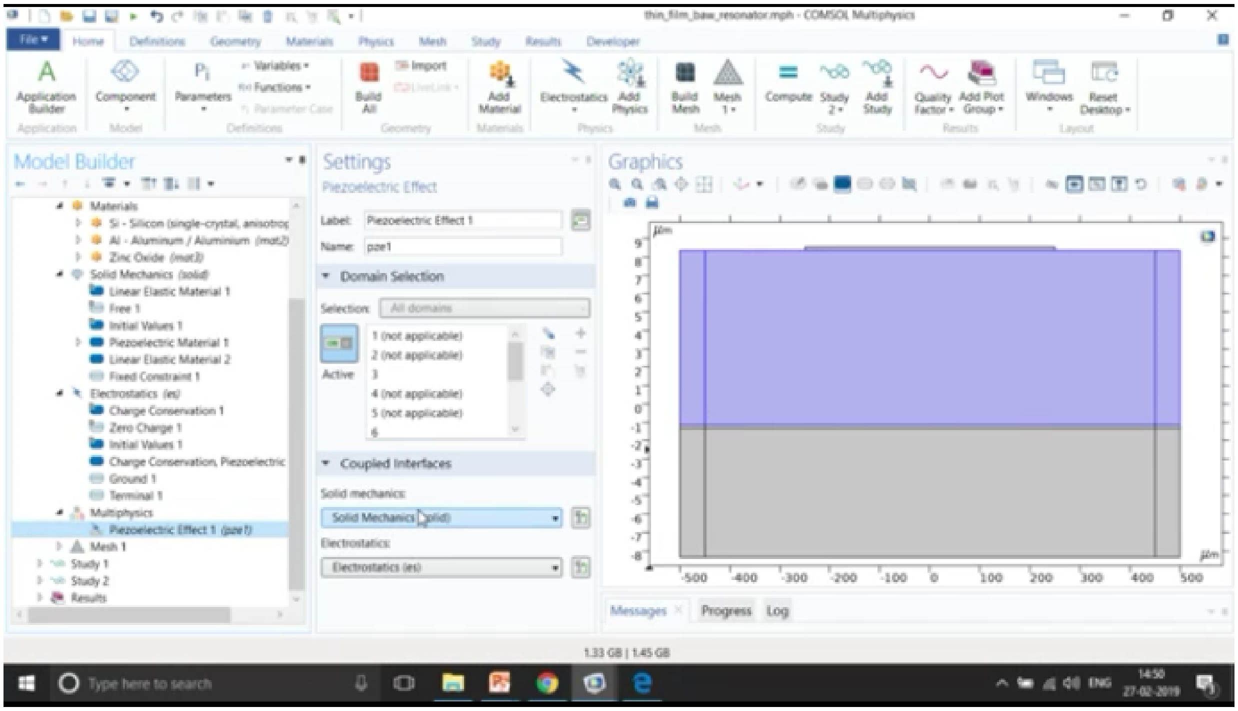
(Refer Slide Time: 06:08)



but we have not selected over here, because we know that the potential within these aluminum domain would be same, the electric field within this aluminum domain is going to be zero, so we use only boundary condition on the edges of your aluminum plates, so here you can see in the bottom plate we have given a terminal boundary condition with a voltage of 1 volt.

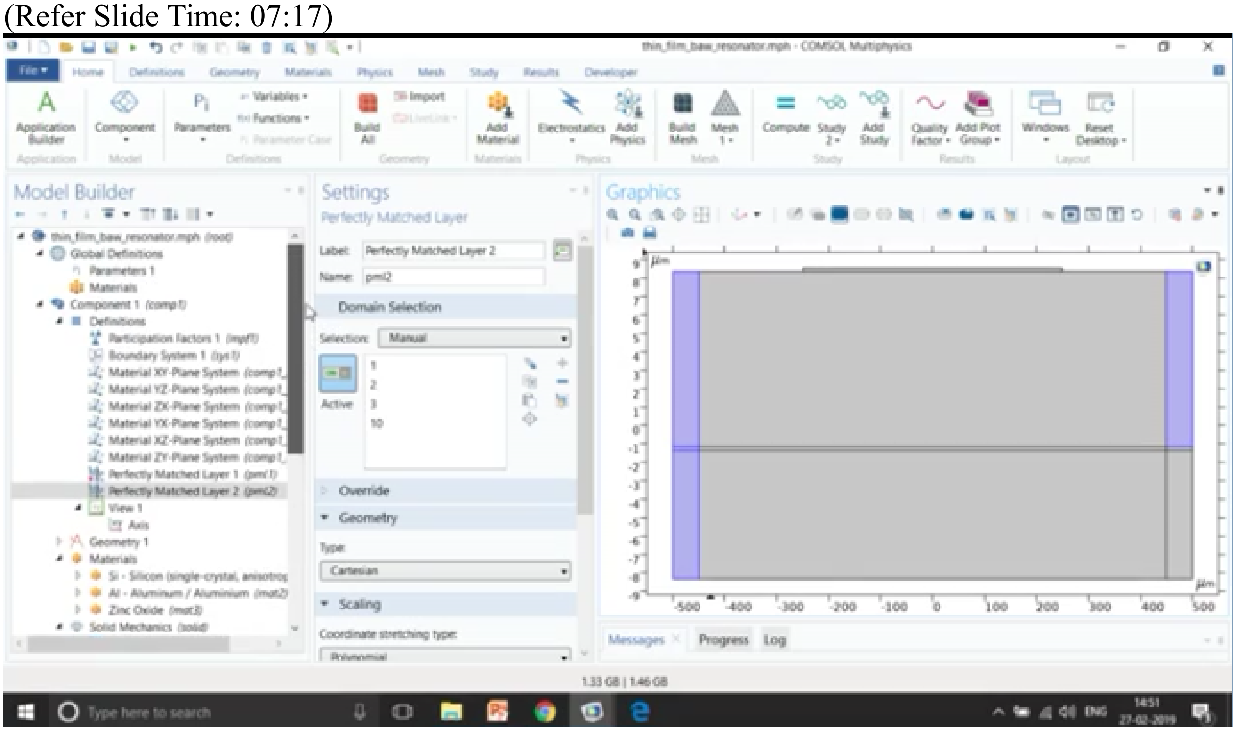
And we have used a ground boundary condition on the bottom of the zinc oxide, okay, and then we have finally used a multi-physics node which actually couples the solid mechanics with the electrostatics,

(Refer Slide Time: 06:42)

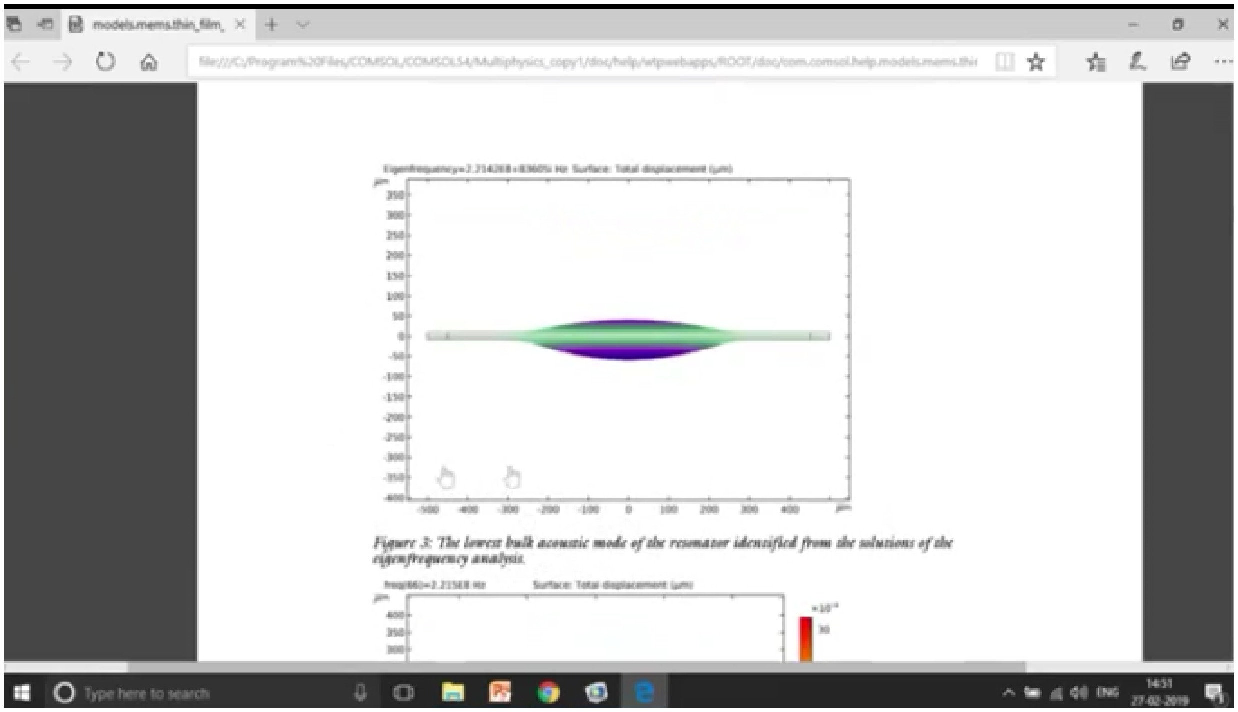


okay, what I was telling before is the use of PML's, so over here if you want to use the PML's you can use it, that's rightly called definitions, and then you go to perfectly match layers and then apply PML's over here, right, so this is how you can actually give the PML's.

(Refer Slide Time: 07:17)

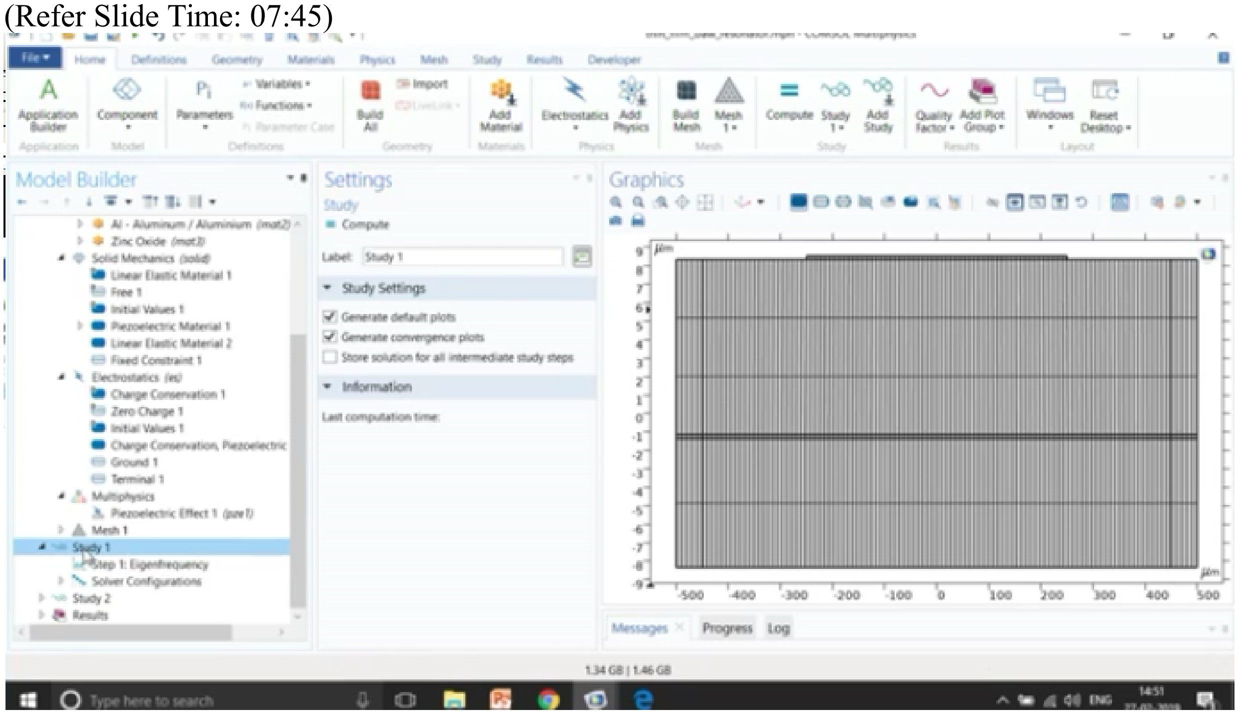


Okay, so let me go and jump to the results part,
(Refer Slide Time: 07:22)



so this talks about the different acoustic modes, okay, so before results I need to show you what kind of analysis was performed, before analysis the mesh part, so in the mesh you can see a mapped mesh has been performed over here, very simple mapped mesh, if you want you can go with the physics control mesh also, but the mapped mesh is more structured,

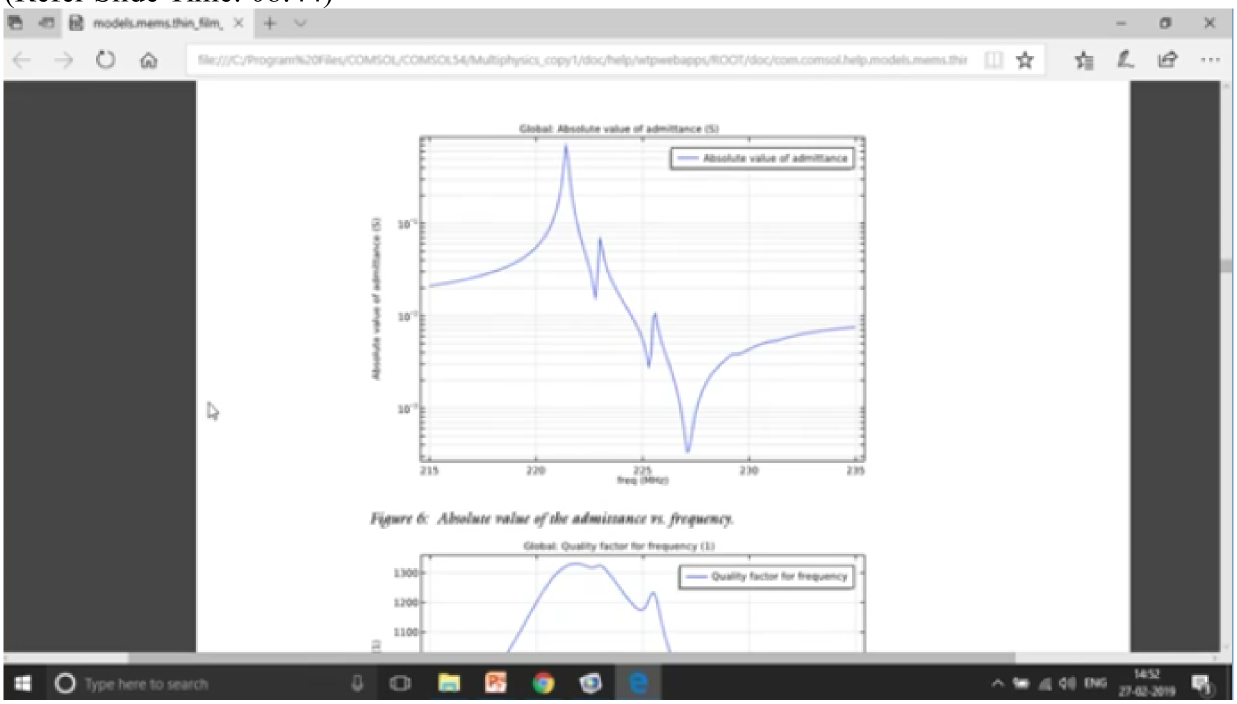
(Refer Slide Time: 07:45)



and then the first thing that you 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.

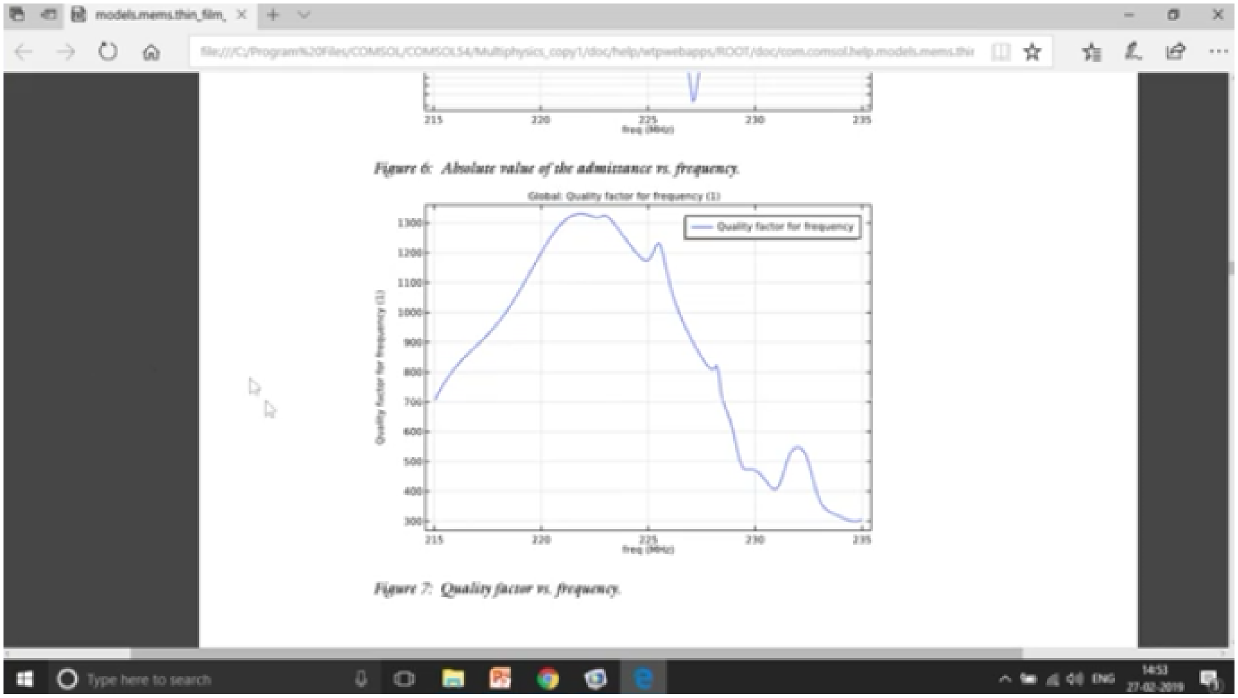
And once you know that your eigen frequency is around 220 megahertz, you then do a frequency domain sweep, so you sweep the frequencies from 512 megahertz to, sorry, 215 megahertz to 235 megahertz with this type of 0.1, and then see as parameters if you want, you can see the admittance plot, all the 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 results, so this is the different resonant modes of your bulk acoustic waves, and this talks about the potential drop, and then this one talks about the admittance versus frequency,

(Refer Slide Time: 08:44)



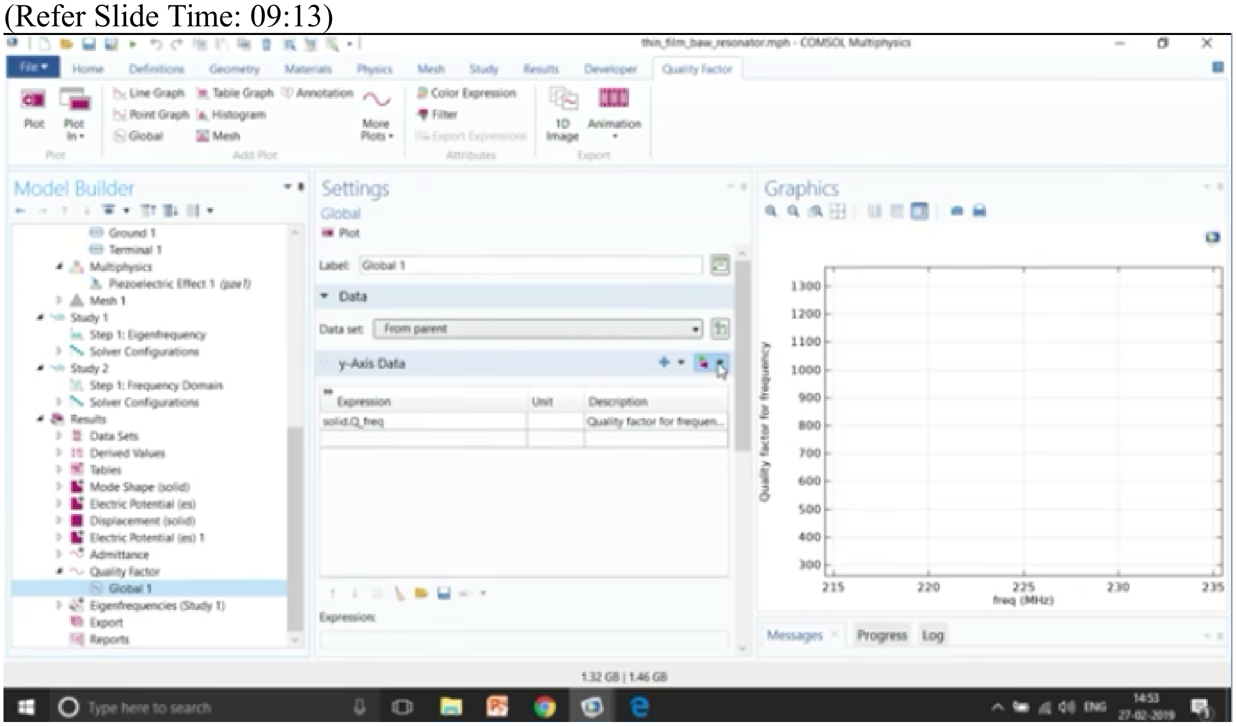
so here we do a sweep of the frequencies from 215 to 235, and how does the admittance look with the sweep of the frequency, we call it the factor which tells you about the how sensitive your system is,

(Refer Slide Time: 09:00)



can also be calculated with a direct formula, so here with, for quality effect, this is an internally defined variables, so you don't need to search for that,

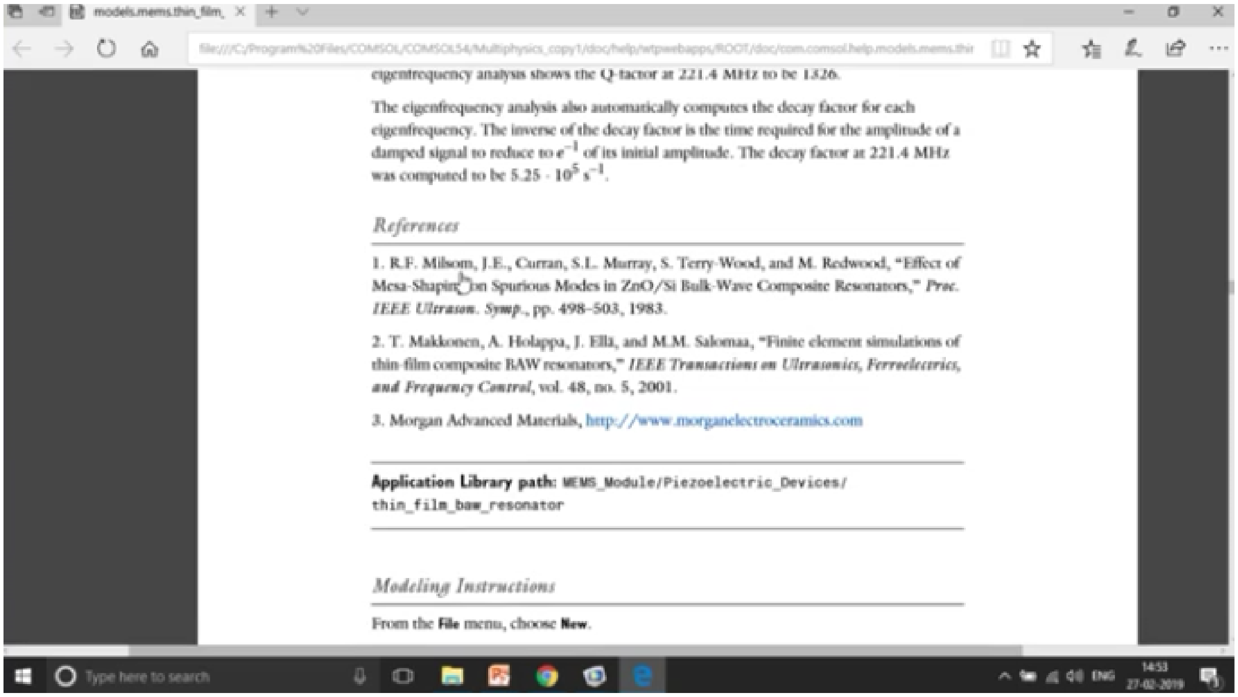
(Refer Slide Time: 09:13)



so you just search for expression, and just write quality factor, okay, for that you need to choose this thing Q, okay so once you solve it at that time you will get it, you just need to search over here the quality factor and this variable will automatic come into picture, okay.

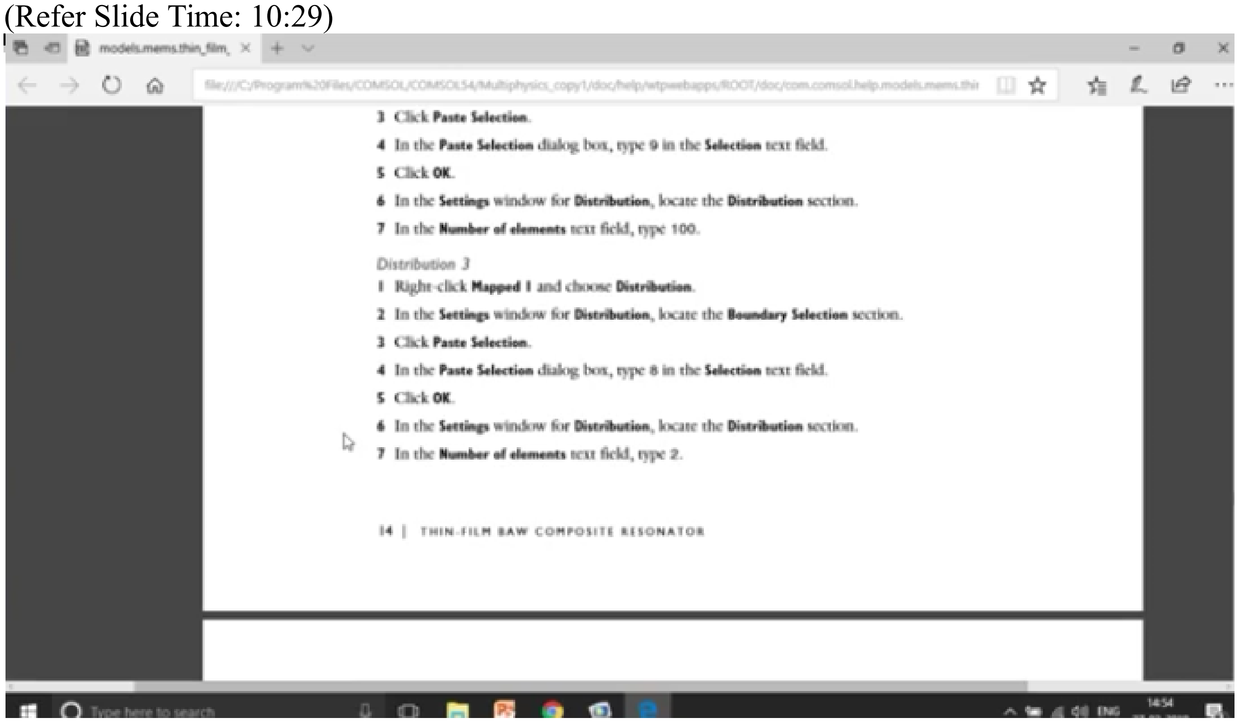
So you have the quality factor, and again there are differences with which the results,

(Refer Slide Time: 09:46)



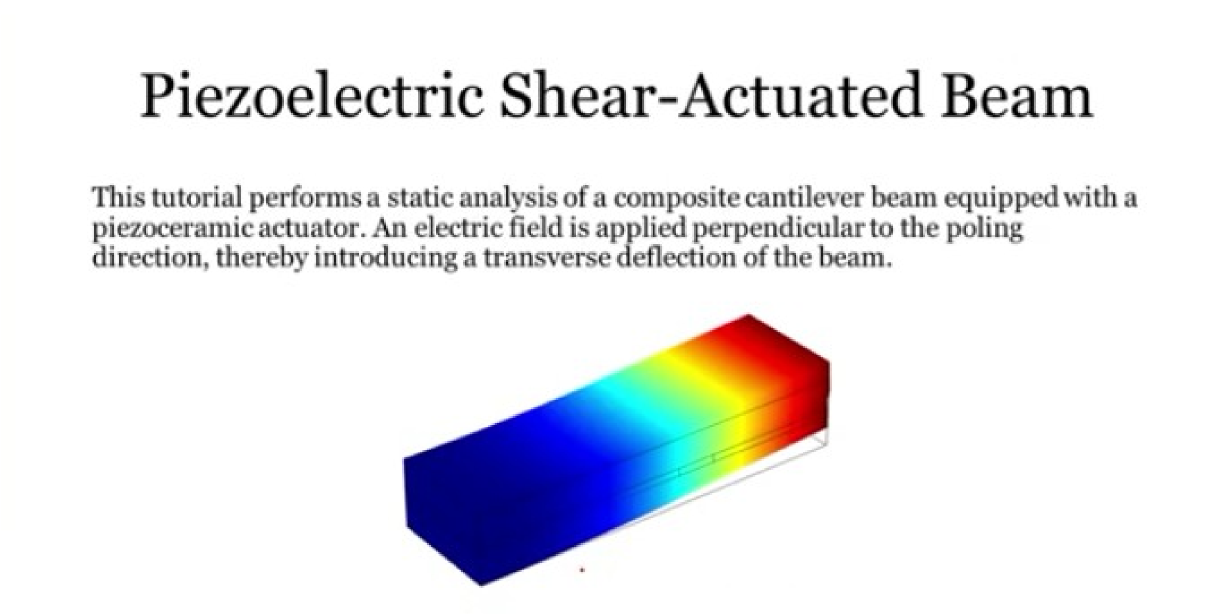
the model has been compared with, so I would strongly recommend you that once you do this simulation you also go through this references to know how this model was set up and what are the material properties and why those material properties more importantly, and then you have the step by step process to make the whole model, 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, once you are done with this particular model then you can change this example model based upon your requirement, and then see the admittance plot,

(Refer Slide Time: 10:29)



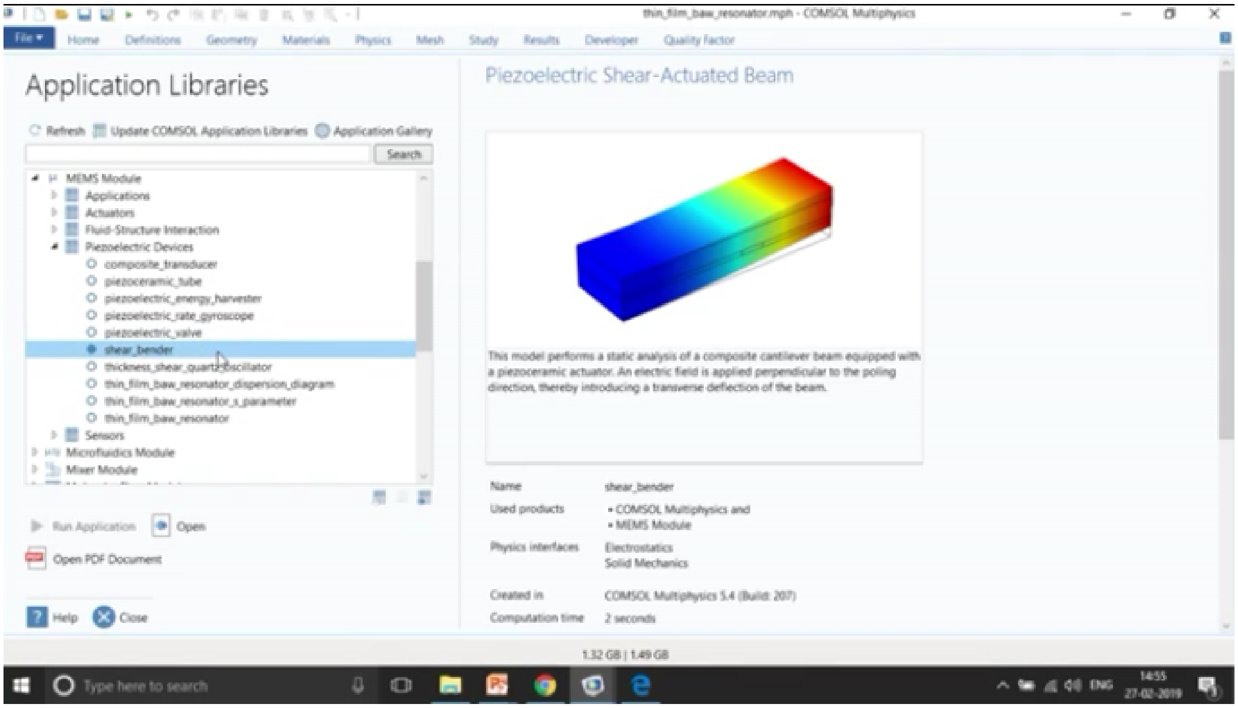
the S parameter plot and other analysis.

(Refer Slide Time: 10:37)



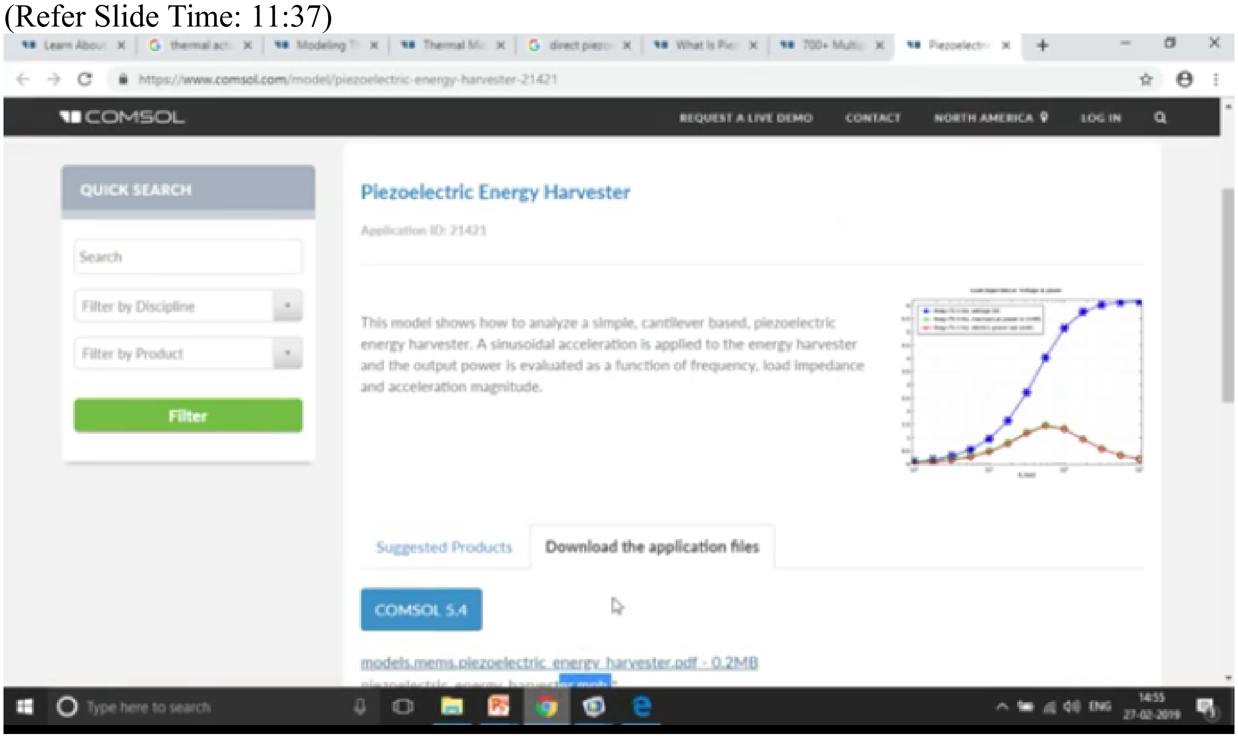
Okay, so let us go ahead with one more example of Piezoelectric Shear-Actuated Beam, so in this example what we are going to do is, we are going to have one piezoelectric device in the middle which is covered with the foam on the left and right side, and then we have two structures, elastic models on 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.

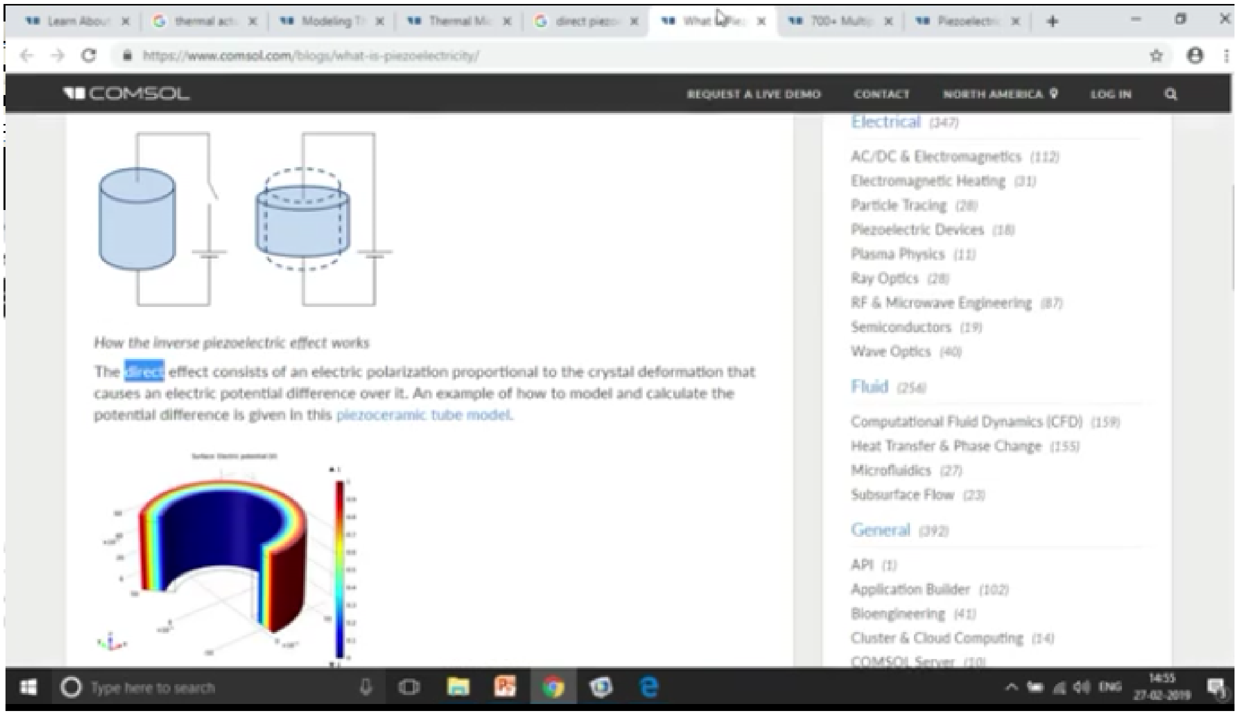
So let us go and open the model, so I just go to COMSOL, I go to file application library, and over here I search for shear bender, so this is the example model, this is again a very example,
(Refer Slide Time: 11:24)



very nice example model because it talks about the poling direction being different as what is given from the pi default.

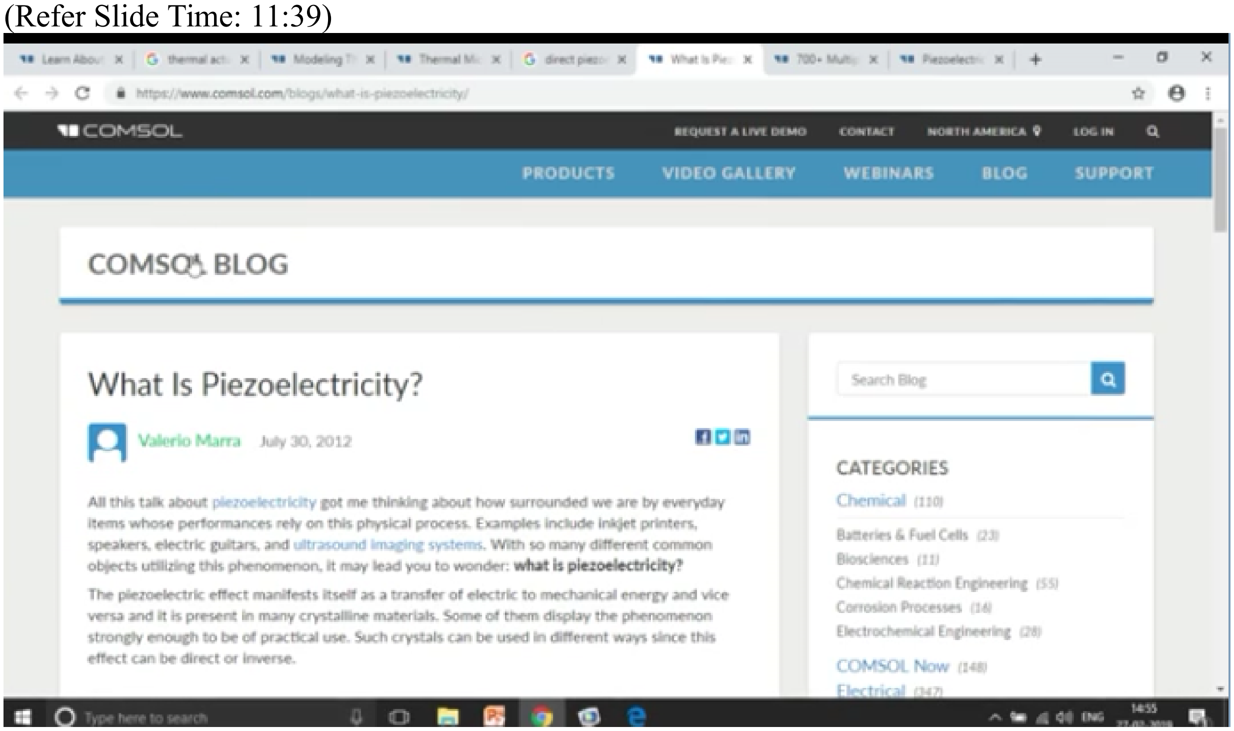
(Refer Slide Time: 11:37)





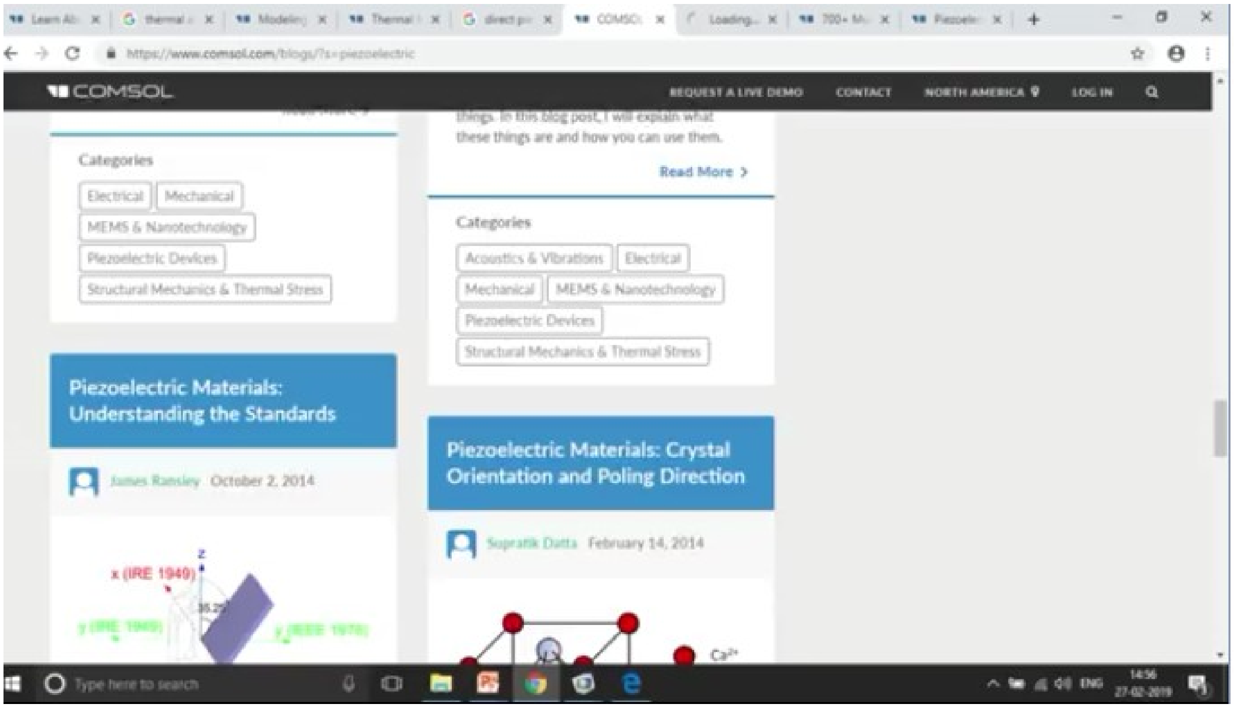
If you want to know about the poling directions, so I would recommend you to go to the COMSOL blog, so which is a COMSOL blog,

(Refer Slide Time: 11:39)



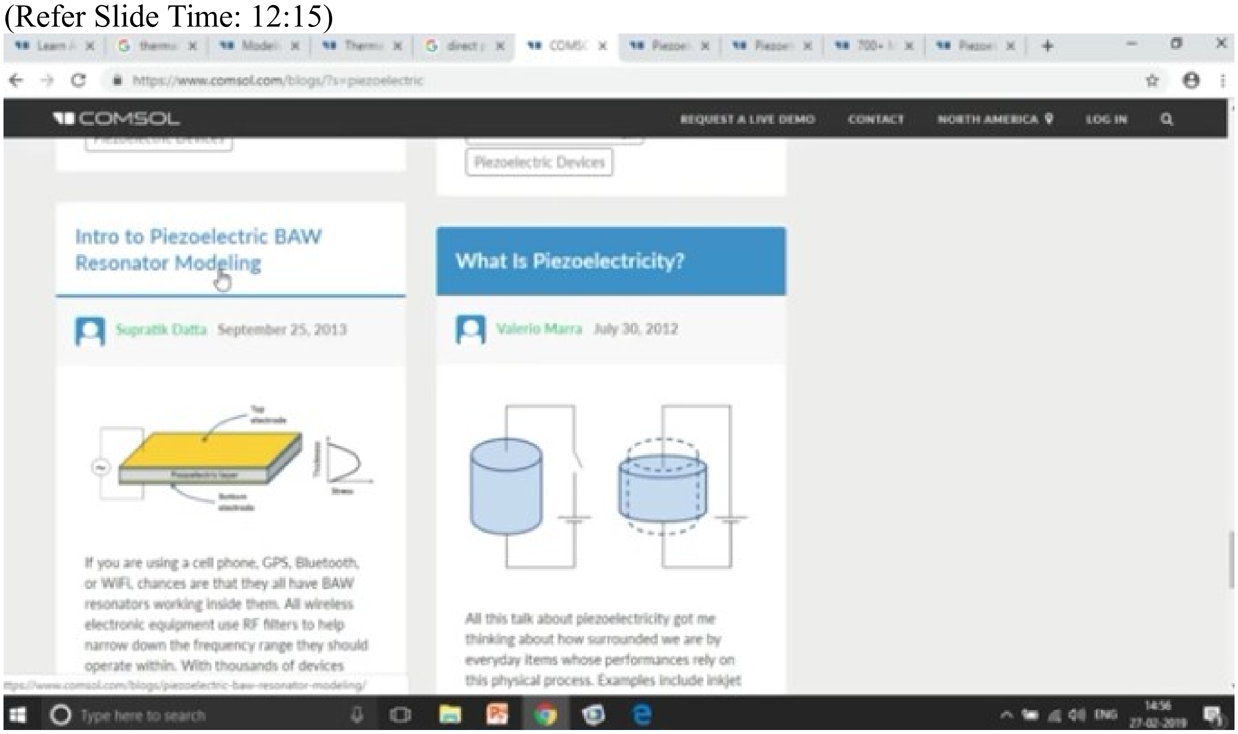
so if you go to comsol.com, and then go for the blogs, yeah, so we have many models on piezoelectric devices, the one that I'm telling about is, yeah, one of the example is understanding the standards,

(Refer Slide Time: 12:00)



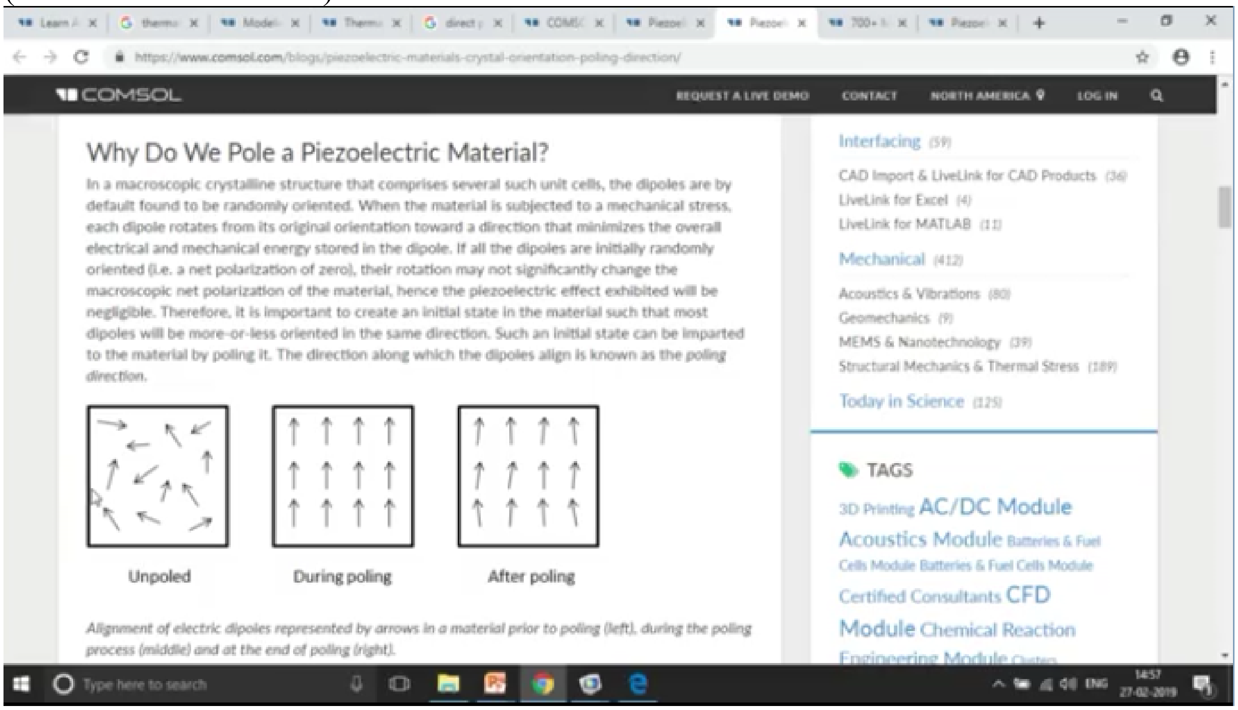
so there are many different standards of piezoelectric materials available, but which standard should you actually go through,

What are the different poling direction, what are the basics of poling direction, these also an models blog for resonator model right, so you can also see these blogs,
(Refer Slide Time: 12:15)



because blogs are very important because they explained this in a very simple manner, how the fixed 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, so I was talking about the poling direction, so a piezoelectric material along with this orientation and poling direction, so any particular material will have its own poling direction based upon where this atom which is defining that poling direction is associated with,

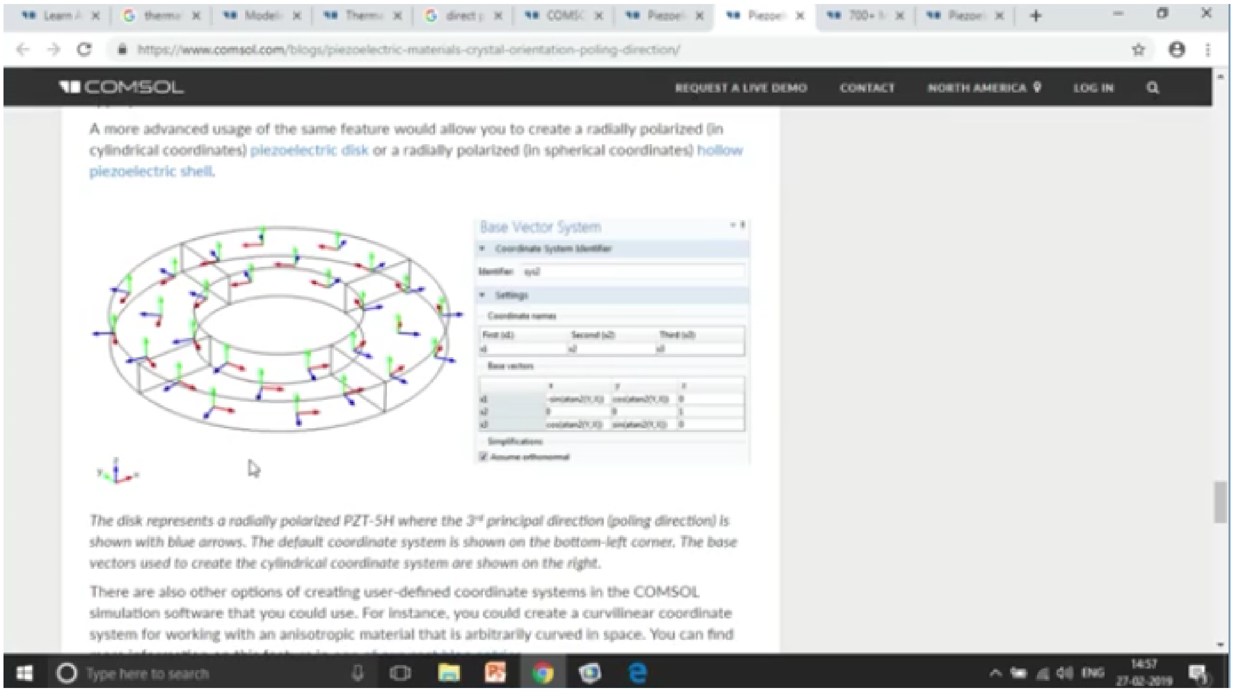
(Refer Slide Time: 12:54)



and once you give a particular external electrical field you will see the dipoles to be arrange in a particular direction quite, so during the poling it switch a particular direction that is along the electrical field, but if you cut a particular crystal in a particular angle, then the poling directions could be updated, right, so if you want to give a particular angle to the poling directions you can actually use a rotated systems, right.

If the different, different angles that you can see over here, different material properties, so if you want to use out, which is radially outwards,

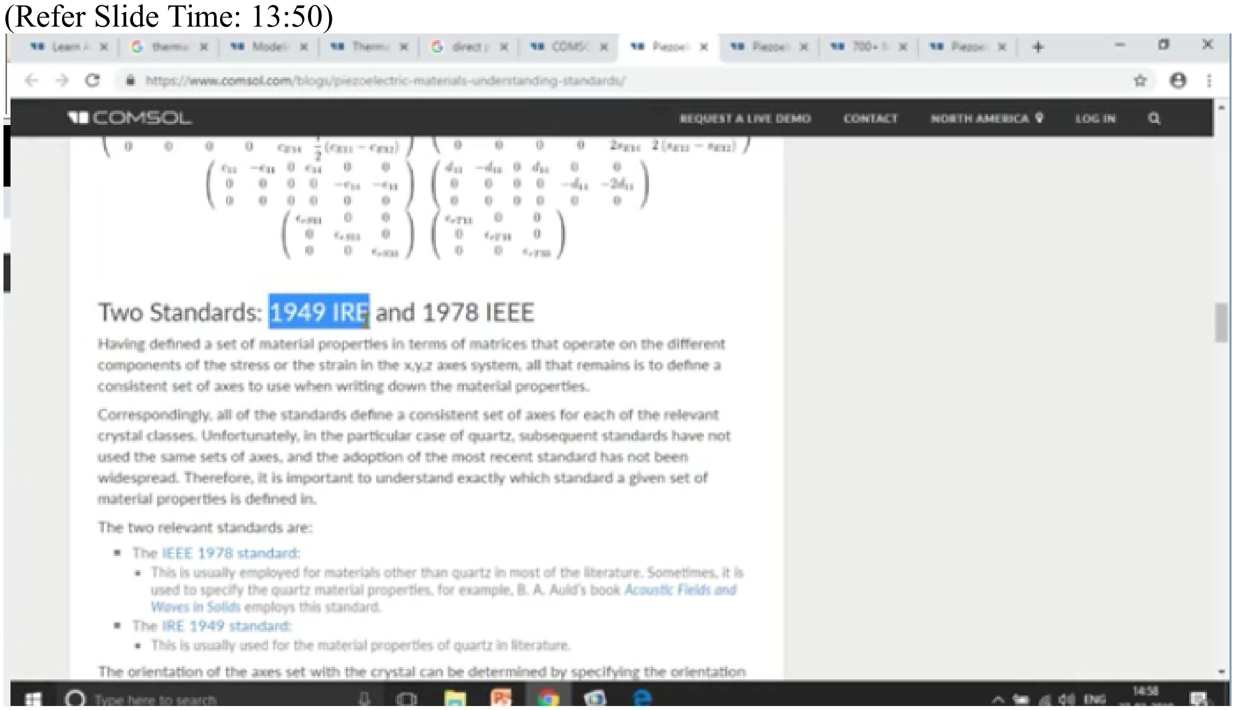
(Refer Slide Time: 13:31)



while a poling direction which is radially outwards you can define those particular types of coordinate systems and then assign the piezoelectric material based upon this particular coordinate systems, radially outwards.

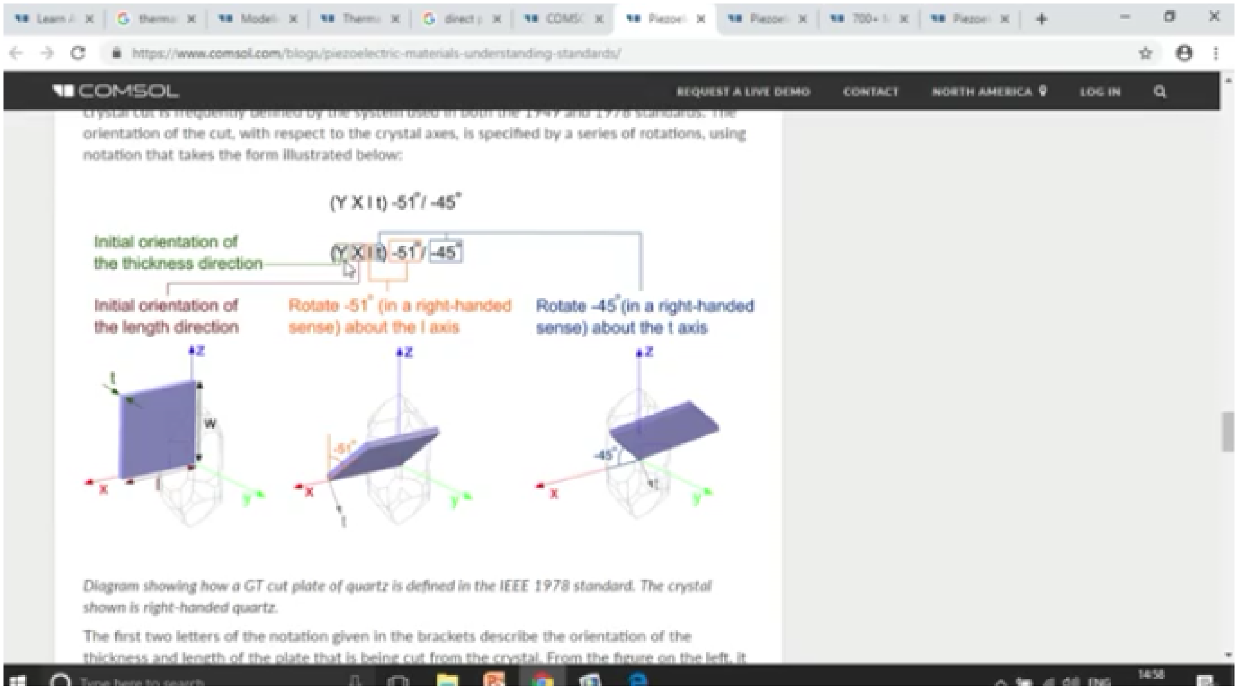
Then we have different standards of piezoelectric materials right,

(Refer Slide Time: 13:50)



so basically 1949 IRE and 1978 IEEE, these are two basic standards for piezoelectric materials, and those also have a particular kinds of poling directions right with a particular cut which is defined based upon the specific numbers that they have, right,

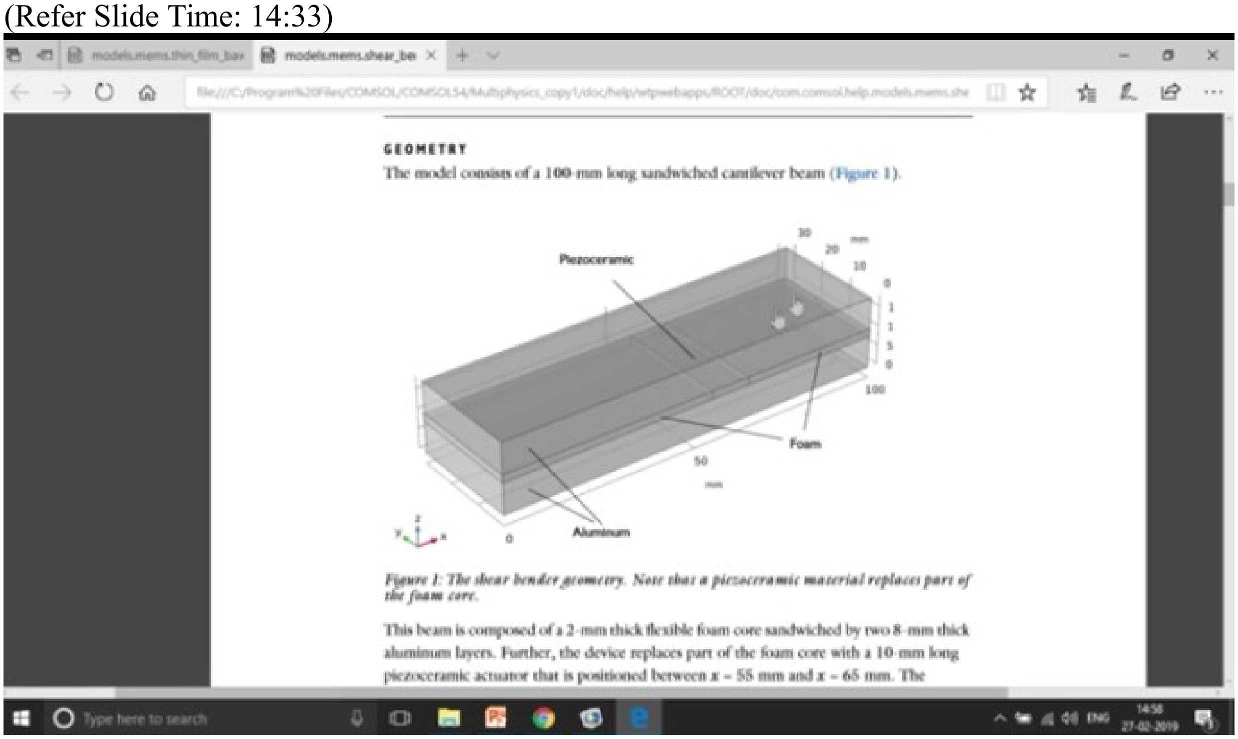
(Refer Slide Time: 14:08)



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

So let me just go ahead and open this particular model, 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 piezo ceramic domain in the top,

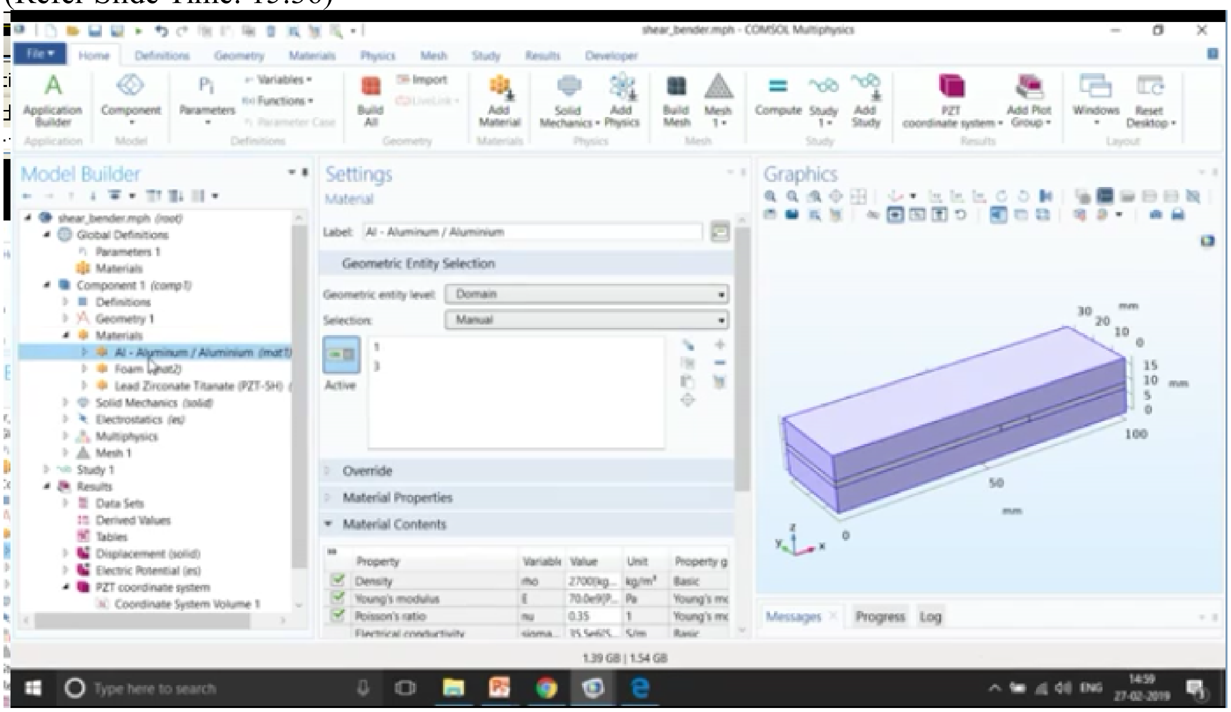
(Refer Slide Time: 14:33)



and we have the same, sorry piezoceramic is the piezoelectric material, and we have aluminum plate in the top and in the bottom, and on the side ways of the piezoceramic material foam has been applied, so on the left and right of the piezoelectric material foam has been applied, right.

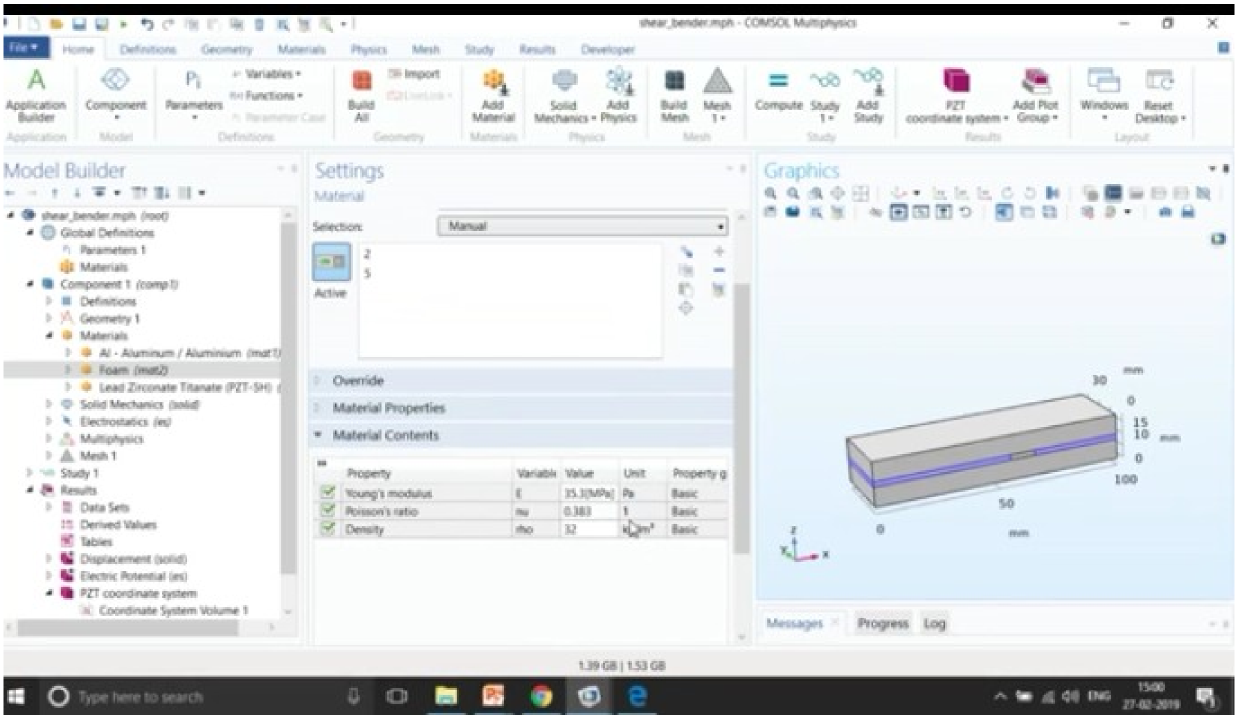
And then we give in the top and the bottom of the piezoceramic material we give a particular voltage to it, so let me just go to COMSOL and open this particular model, okay, so let me go from top to bottom, so in the comprehend section first thing is to make the geometry, so I just go ahead make the blogs and make the geometry, I first make a blog and they are introduced this center domain using layers over here, and then I assign the material properties,

(Refer Slide Time: 15:36)



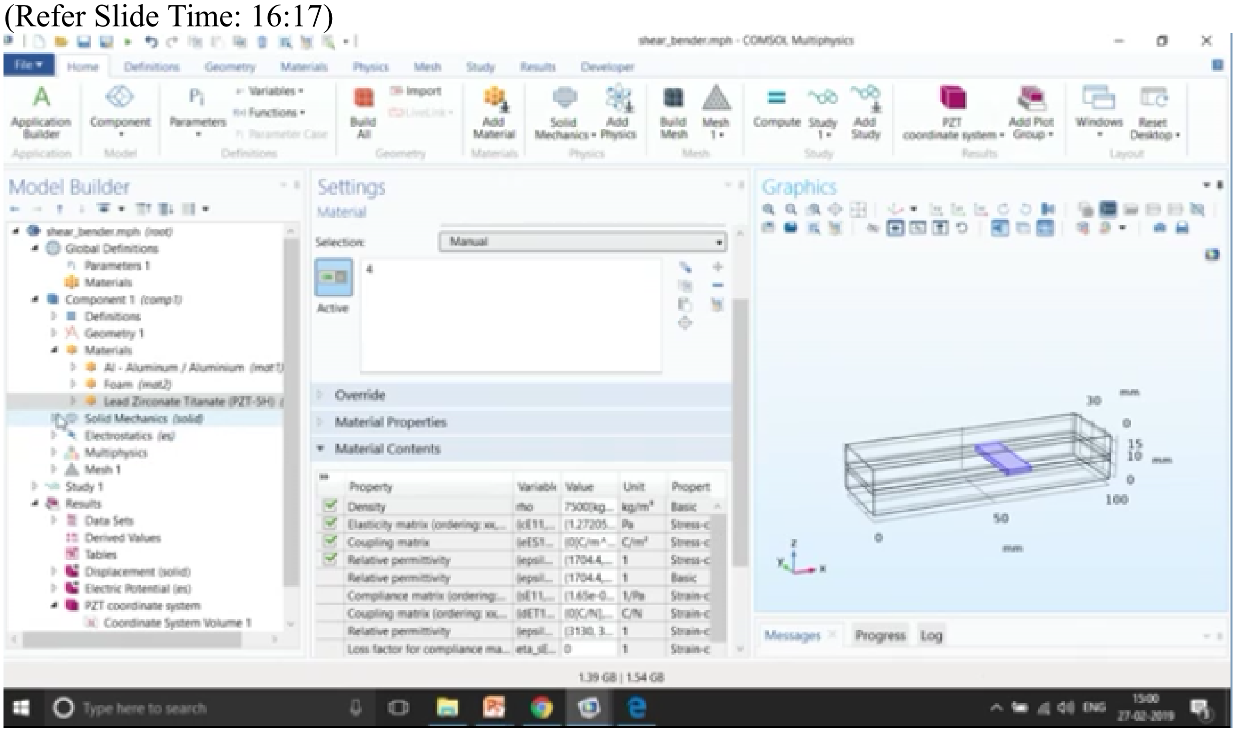
most important, aluminum domain in the top and bottom, you can see the blue part the top and bottom is aluminum domain, and here you assigned it as scalar material properties, okay.

(Refer Slide Time: 15:54)



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 there you can see, and then finally the PZT-5H and here we give a tensor material properties as you can see over here the blue part. If you want to see through the domain you can anyway enable the via from rendering, so you can see it through.

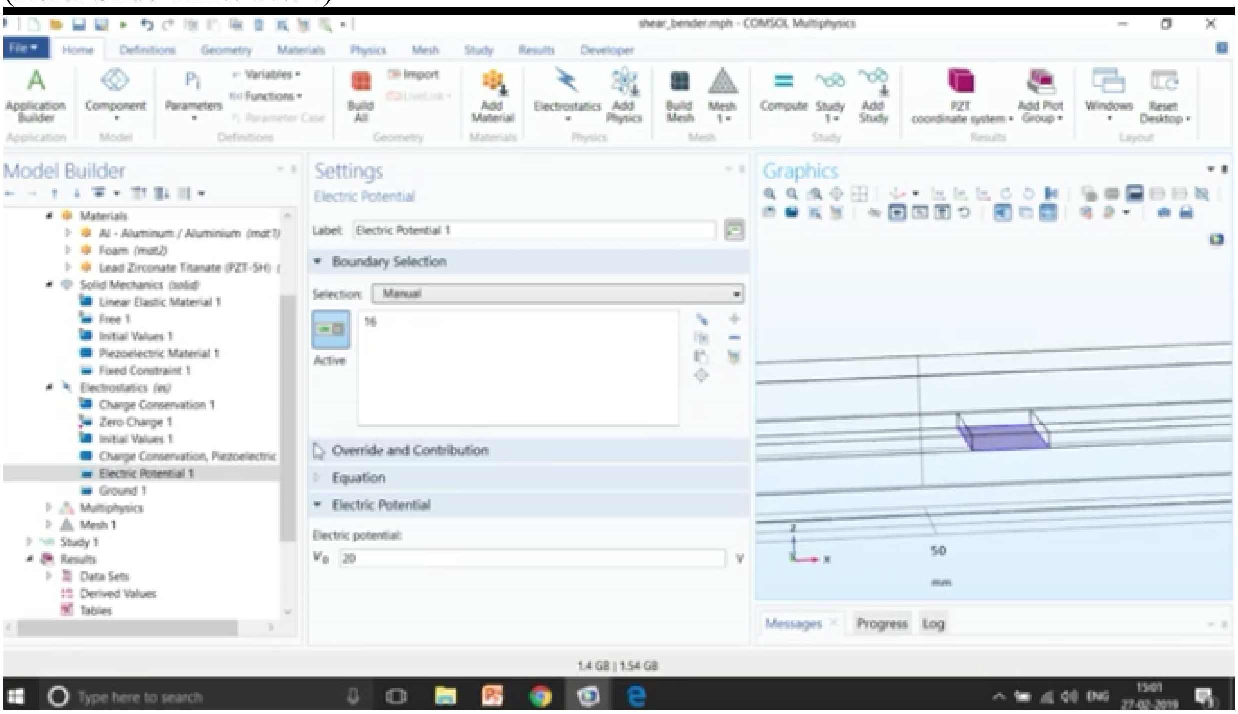
(Refer Slide Time: 16:17)



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

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've given the piezoelectric material property,

(Refer Slide Time: 16:56)

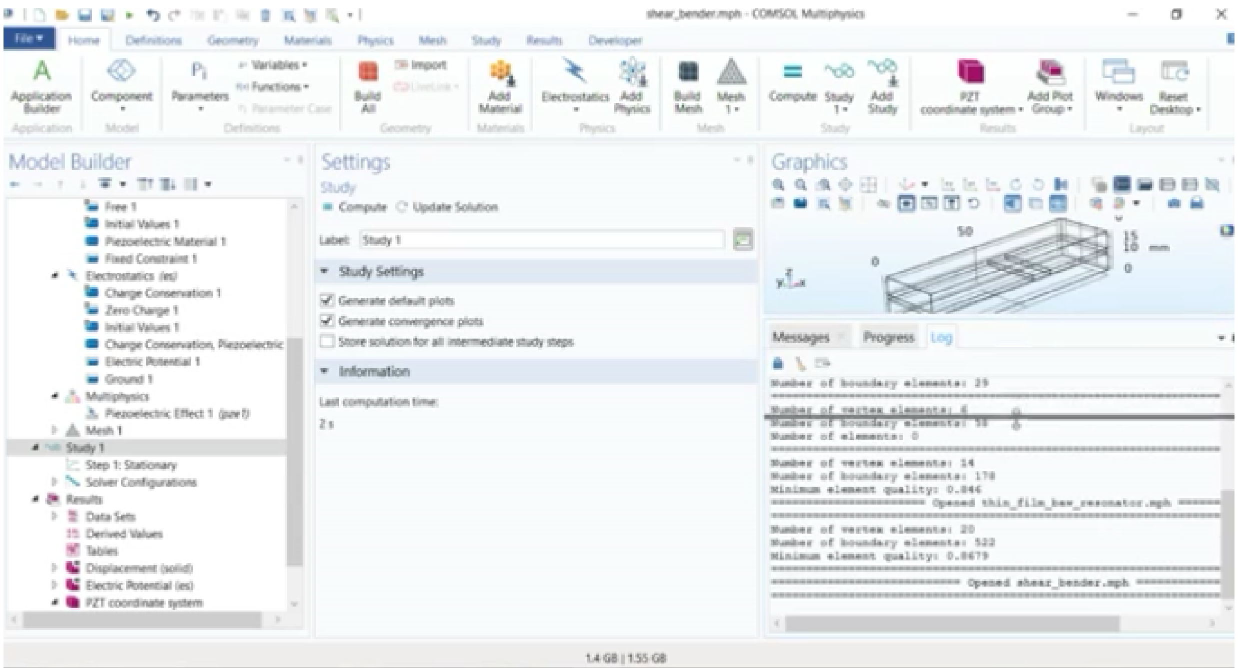


and then in the top part I have given a ground boundary condition.

And then finally to couple both of them I'm using piezoelectric coupling, so this actually couples the solid mechanics with the electrostatics, and then I'm using a kind of a stationary study, so this is going to take only two minutes, right, so you can see how much time it will going to take, it's 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 some time is an example of accelerometer, it takes little bit more time and in addition to the more time it takes you more RAM, right so maybe your system is not having that much of RAM so it is always good to see how much time it's going to take.

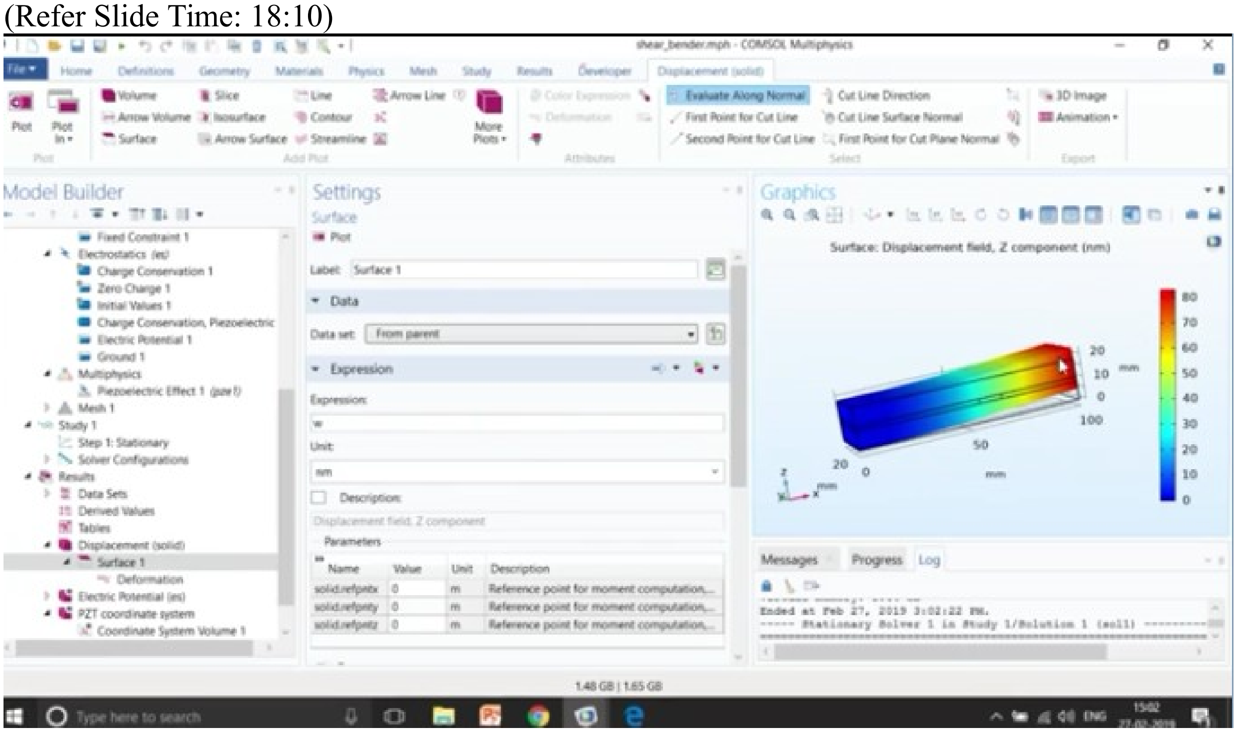
And if you want to see how much time it RAM it takes, I think it should be available in the log values also, so here also you can see how much, it may be showing how much RAM it took to solve this particular model.

(Refer Slide Time: 17:53)



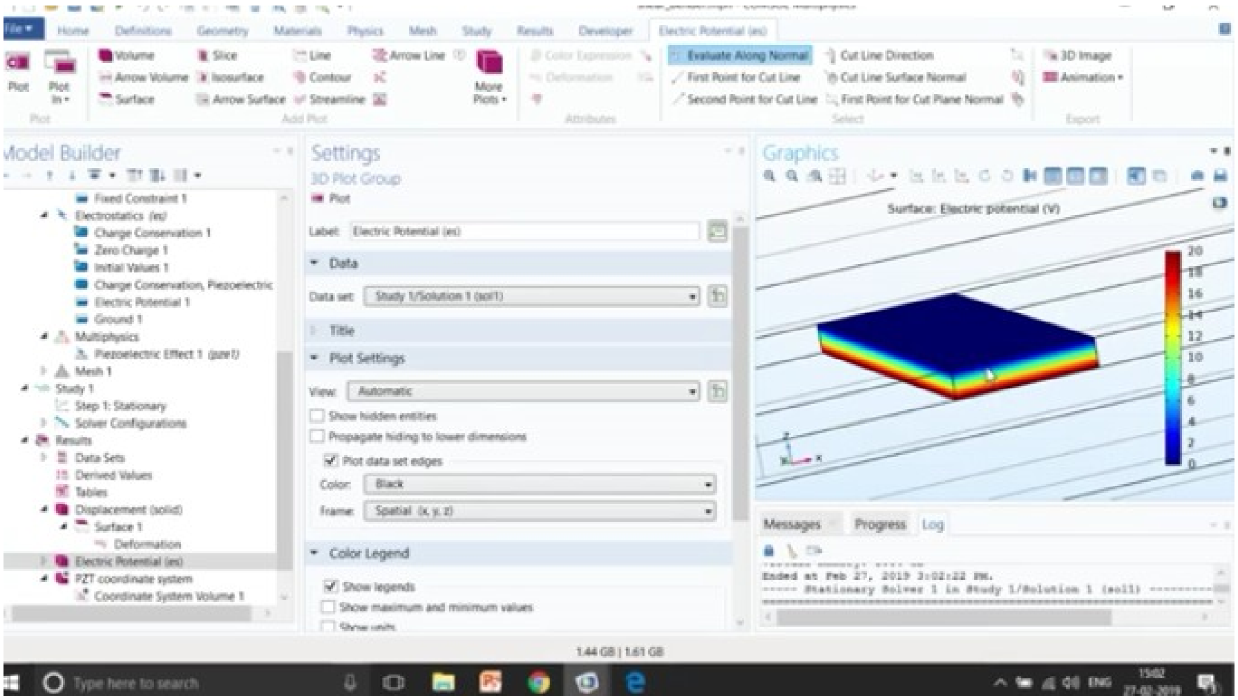
So let me just go to compute, yeah, so now we can see that the displacement which is in order of nanometers,

(Refer Slide Time: 18:10)



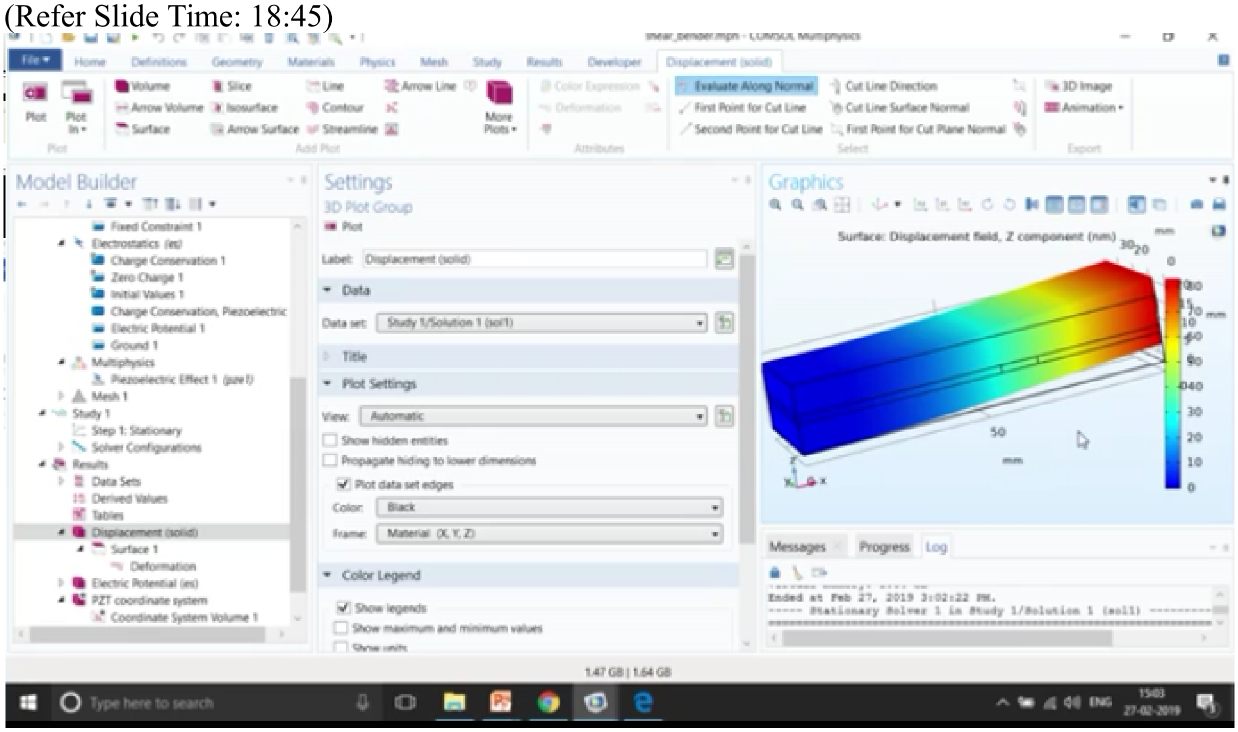
so what do you see right now is an exaggerated view of what is happening within it, the first thing is the electric potential, so we give a high potential in the bottom and ground it in the top,

(Refer Slide Time: 18:20)



right, so you can see a nice deep or deep in sweep 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 of the stresses it's going to deform, so in this case it is as deform in the top part,

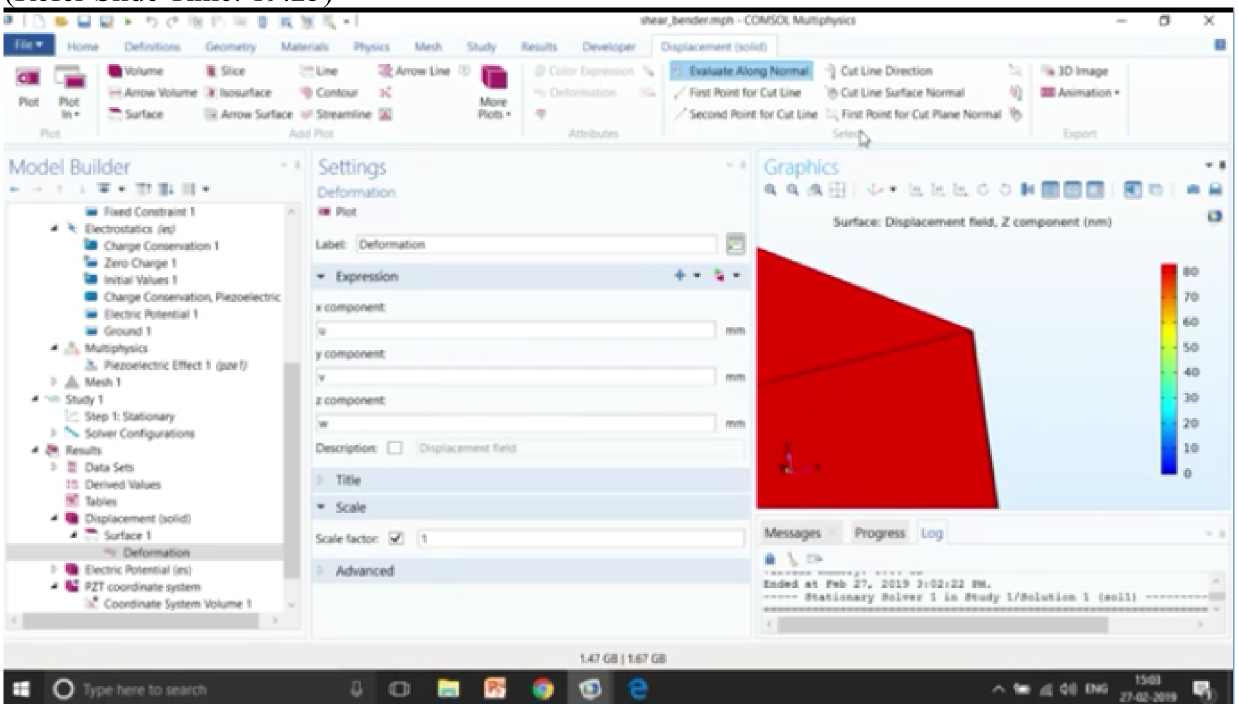
(Refer Slide Time: 18:45)



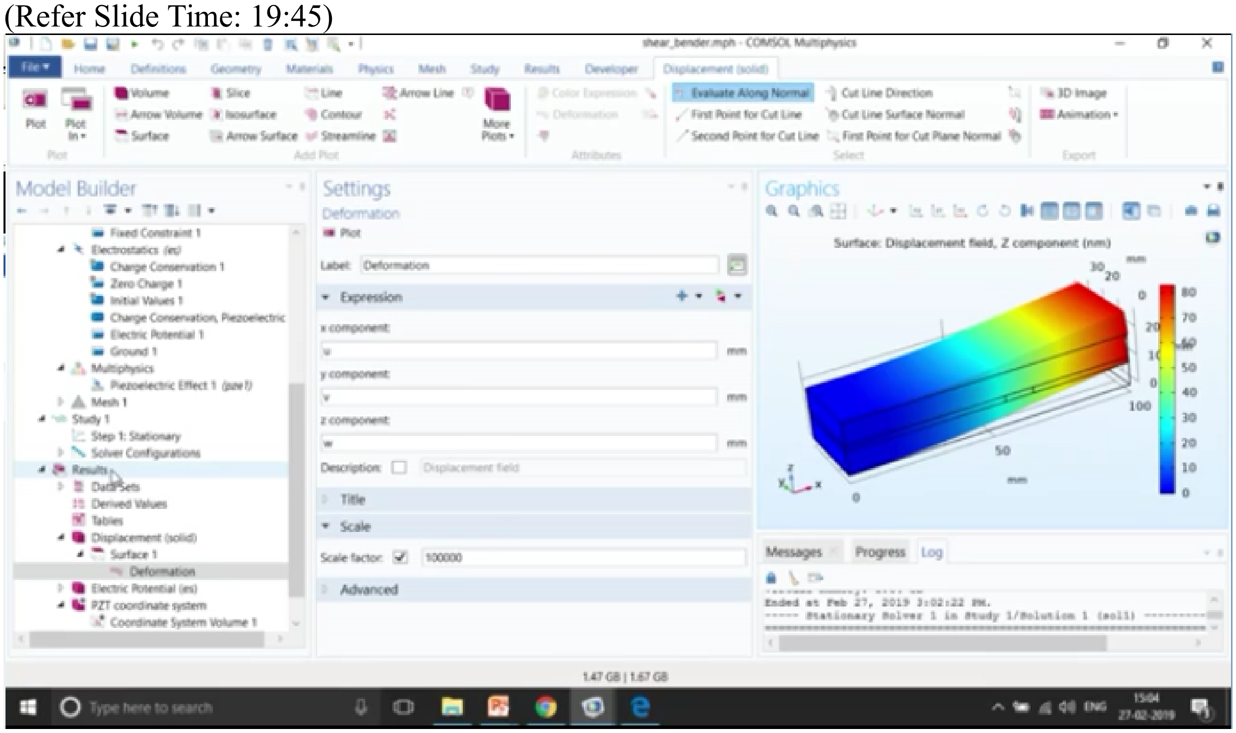
so it's moving top, right, it's moving upwards.

If you 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 have to

go within the surface one to the deformation that you can see over, this in the screen and make it as 1, and then click on plot, so this is the actual deformation, of course and 19 nanometers in a device of MM structure you will not be able to see that much, so if you just zoom it,
(Refer Slide Time: 19:25)



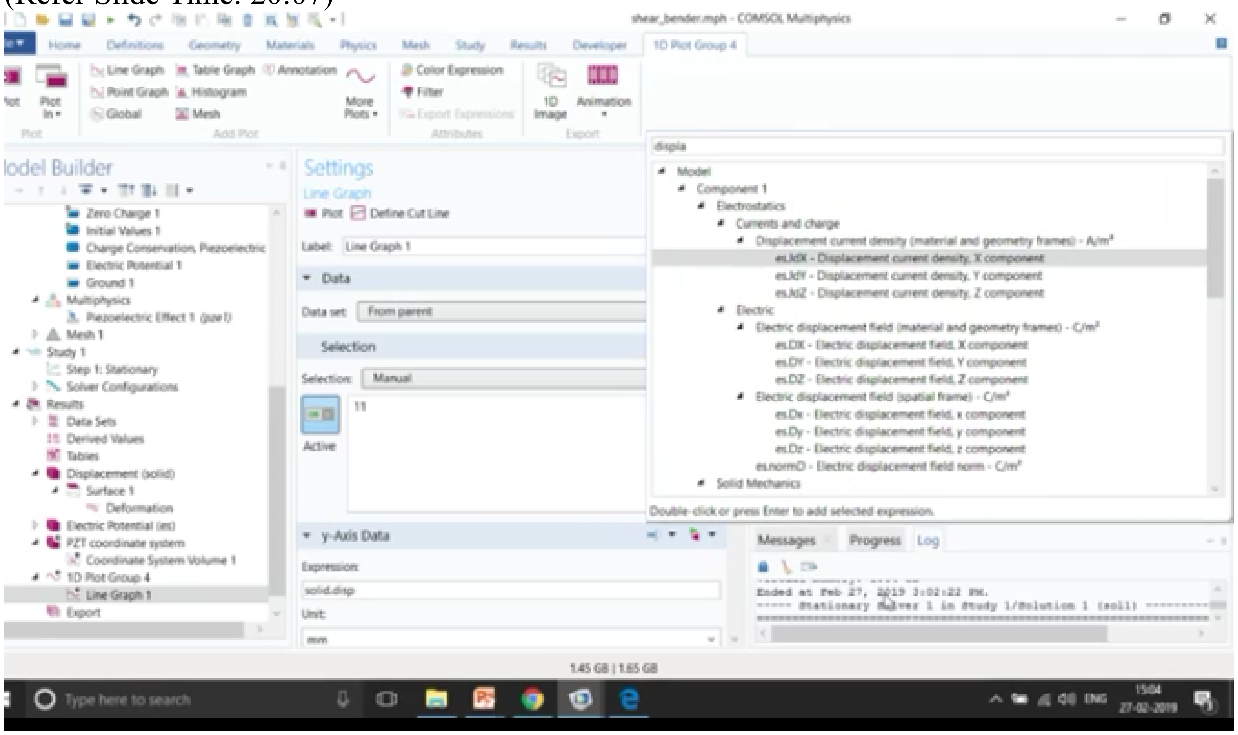
you will not be able to see that much, and that's why we have just scaled it to show you the effect of the deformation, okay, so for example now you want to see the deformation at this particular line, that is also possible, to do that just right click on the results,
(Refer Slide Time: 19:45)



1D plot group, add 1D line graph and choose which line you are talking about, so I am talking about this particular line, right.

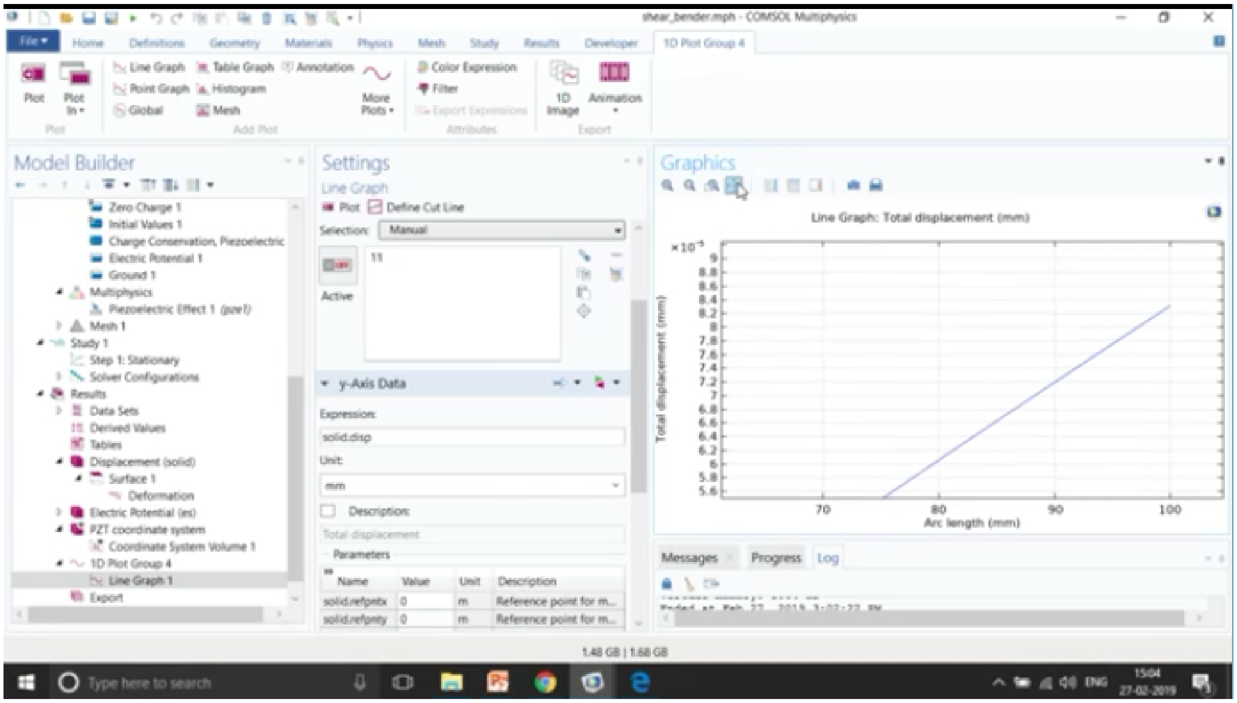
By default this displacement variable has already come, so if you want to search I can also search over here, so I go over here and just search for displacement, and there are many other variable,

(Refer Slide Time: 20:07)



so right now I'm looking for total displacement, right, so I just use the total displacement over here, just double click on total displacement and you will get solid.dispower here, and you can see the displacement, so as you can see from 0 to around 8.3,

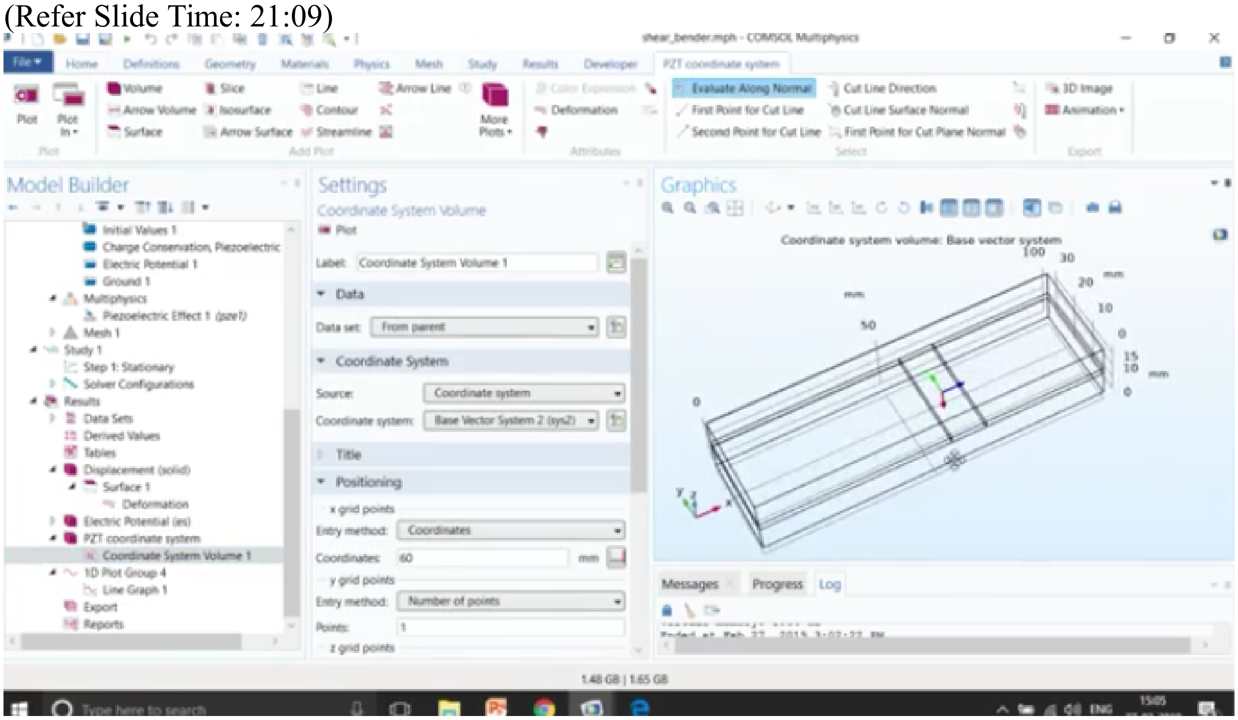
(Refer Slide Time: 20:29)



or around 83 micrometers is the displacement, sorry it's actually nanometer so it's in MM, so let me just go ahead and in nanometers, so it's around 83 nanometers is the maximum displacement along that particular line, okay.

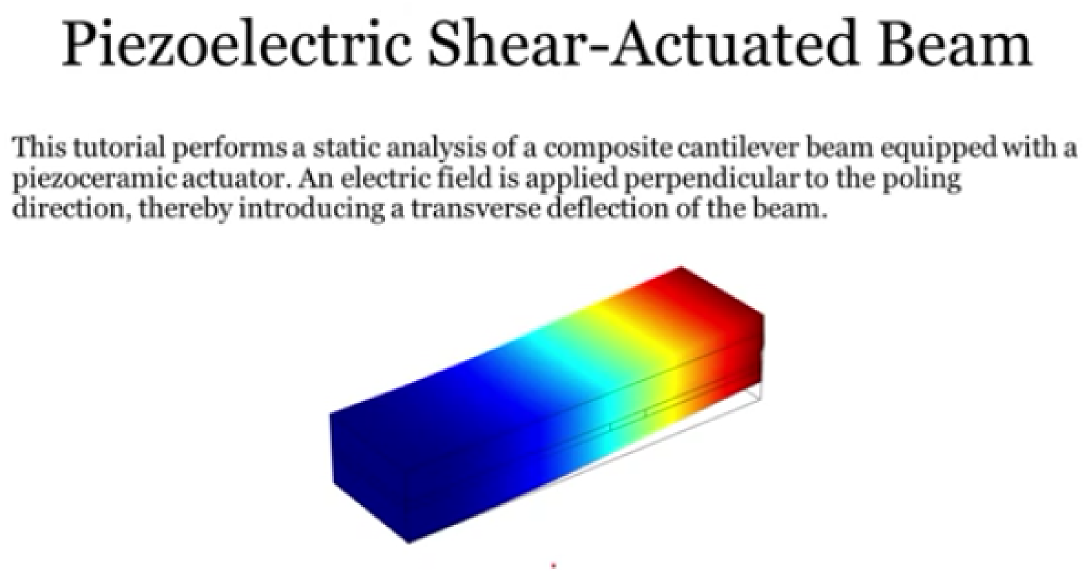
Finally you can see that in this particular case, it's very important example because in this piezoelectric we have used a different poling direction, so by default you will see the poling direction of any of the material properties is along Z direction, but in this case you can see over here it's along X direction,

(Refer Slide Time: 21:09)



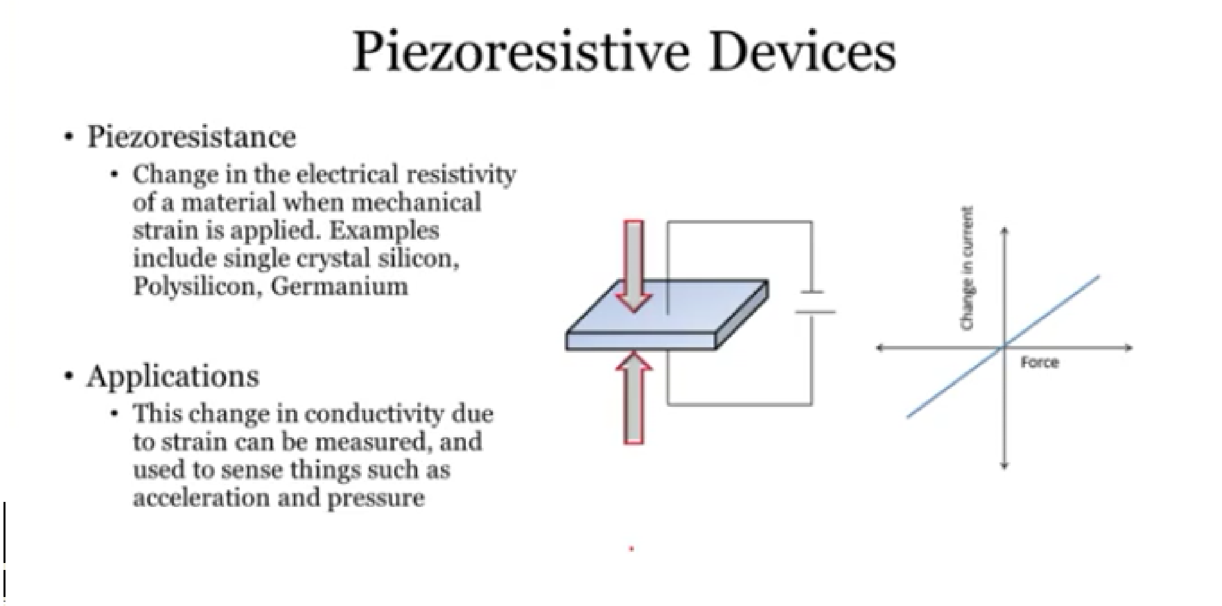
the blue line is around X direction, so blue is the one is the poling direction and that is along the S direction, so the way it was taken into consideration that is the poling direction of in X direction we use in definition, we use a base vector system, and we define that particular poling direction over here in the form of base vectors, okay.

(Refer Slide Time: 21:43)



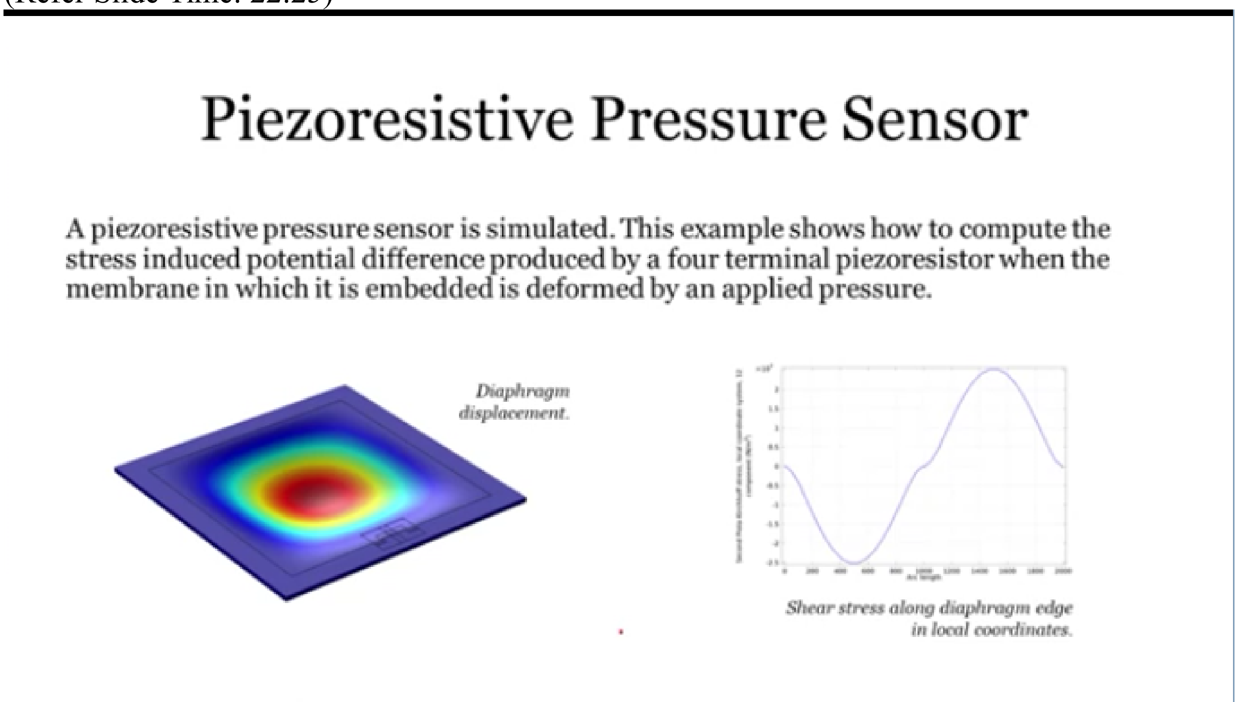
So let us go ahead with the presentation, so this is how the different types of piezoelectric material were model, so let us go ahead with the piezoresistor device,

(Refer Slide Time: 21:49)

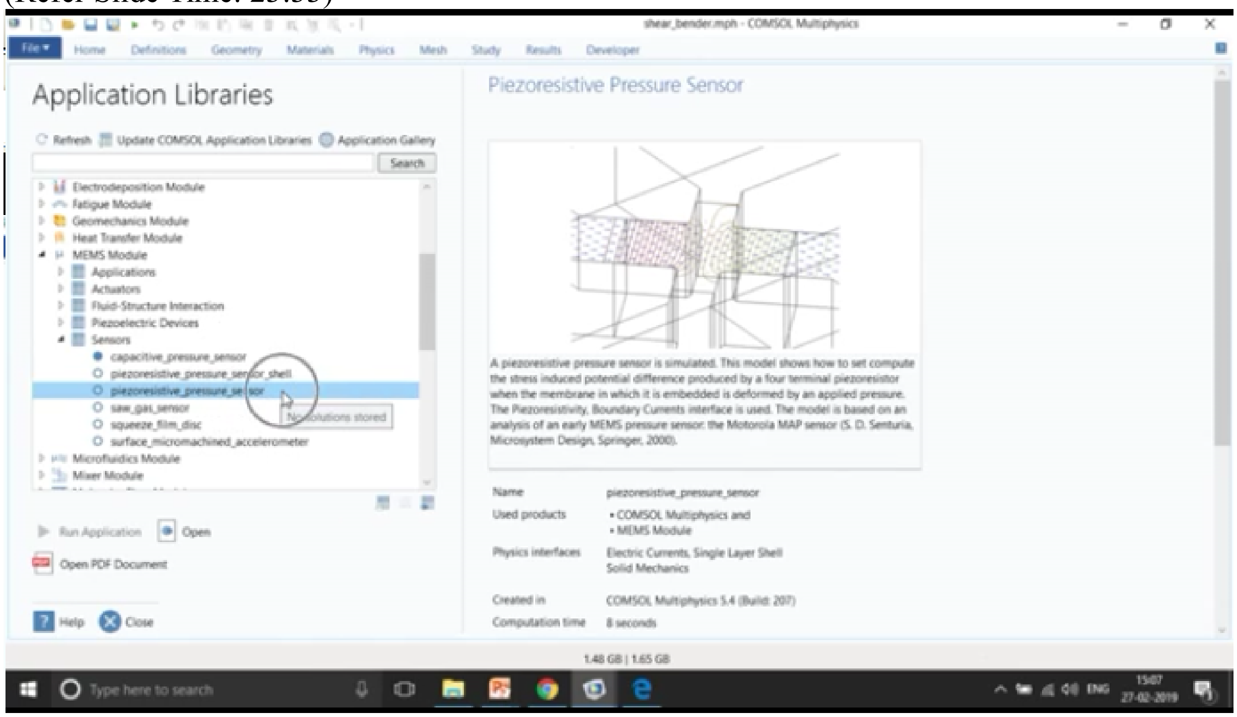


this is a very interesting type of a device because over here based upon the forces that we apply or the stresses that are going to develop it's going to change the resistivity of that particular device, and the way that you can take into account the resistivity as a sensor is to understand how much current that is being withdrawn from that particular system, so in this particular example that we are going to talk about, we are going to talk about how the forces have been 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: 22:25)

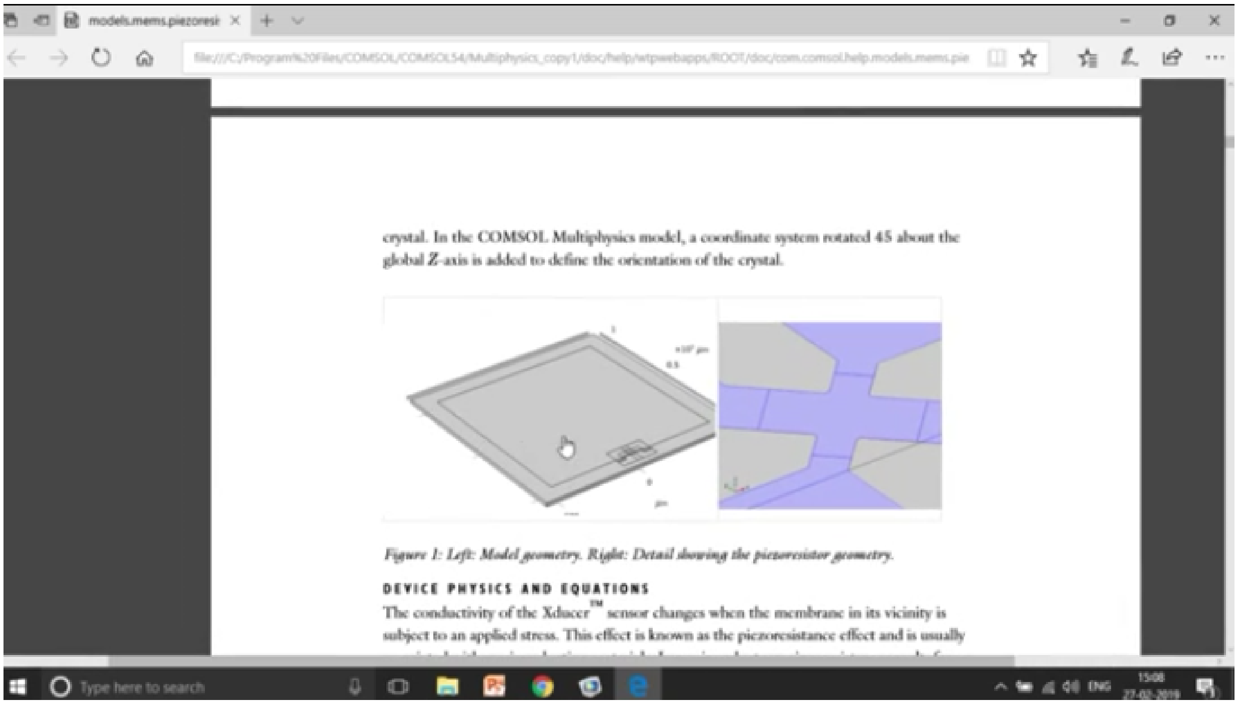


This is the first example that we'll talk about, this is an example of a diaphragm that we can see over here, and we apply a particular force in the top from here, and we have a piezoelectric, sorry peizoresistor material in this case over here with different types of doping, so N doping and P doping both we have over here, and then we try to understand along particular arc in this case how the stresses are being developed, so you can see both the negative and positive part of the stresses that is getting into picture and because of that you can understand how much current is going to be getting placed, so we'll just go to COMSOL again, so I go to file, application library, and I search for piezoresistor, okay, so it would be somewhere over here I guess, yeah, so there are different ways of modeling again, one is a volume metric model that you can see right now,
(Refer Slide Time: 23:33)



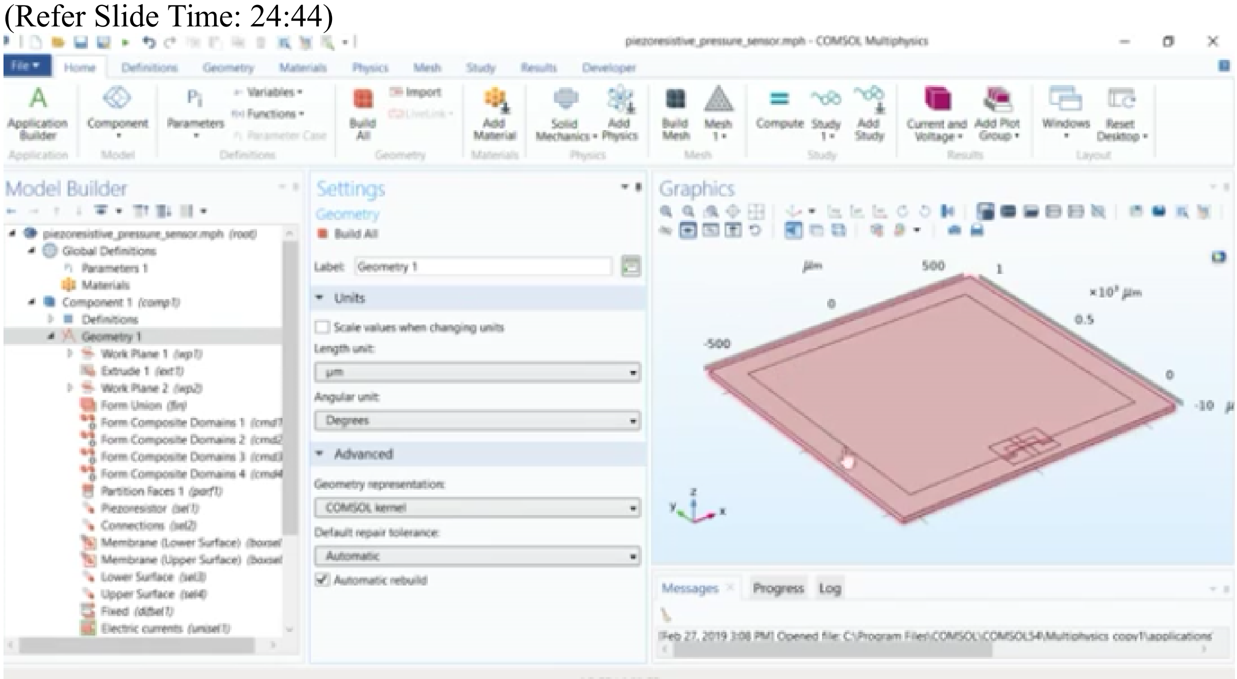
another approach is a shell approach that is modeling only 2D structures, so this is again a different, so it's kind of approximation that we do to quickly get the results which is, the results which are not that far away from the actual results.

As of now let us go for piezoresistive pressure sensor , let's open the documentation, so this is the actual structure that you can see over here,
(Refer Slide Time: 24:00)



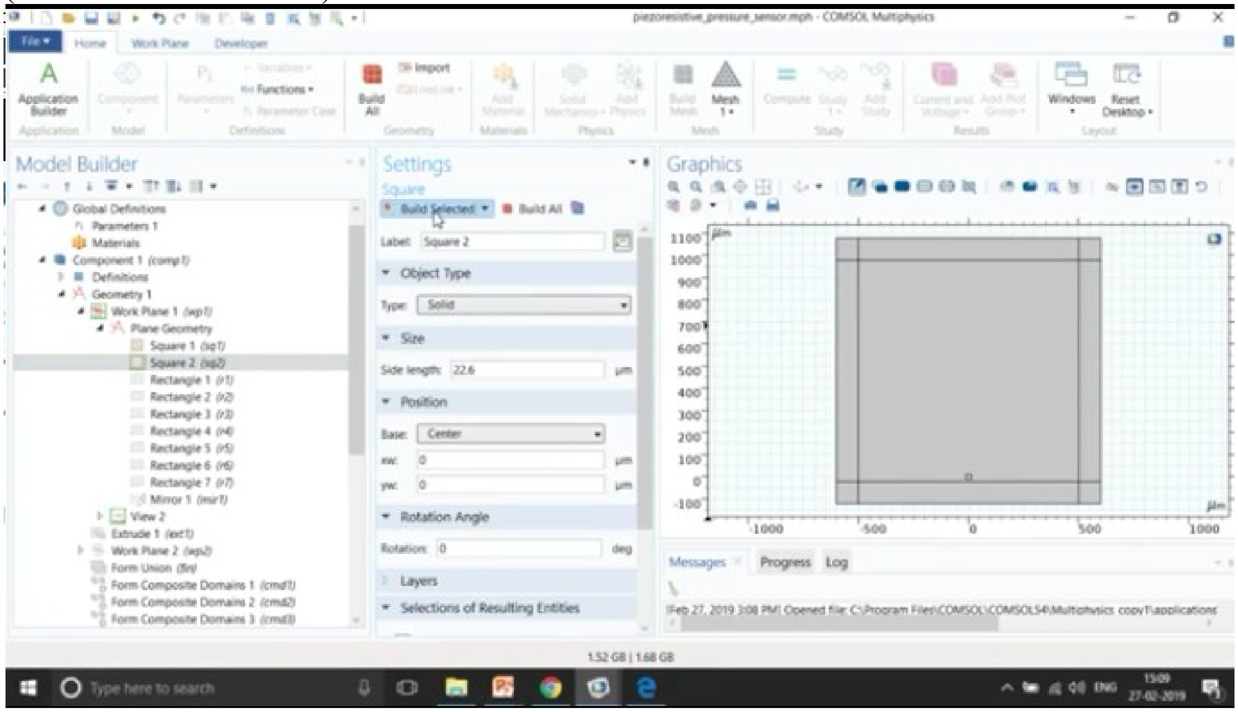
this is a diaphragm over here, and then we have the piezoresistive structure over here which is a kind of V stone bridge kind of a system where we give a particular voltage on one side, and ground on other side and the other two flanges that you can see over here that is used for decreasing the error rates. Again we use a tensor for both elasticity and coupling matrices, and so let us go for the modeling part before we go to the results, okay.

(Refer Slide Time: 24:44)

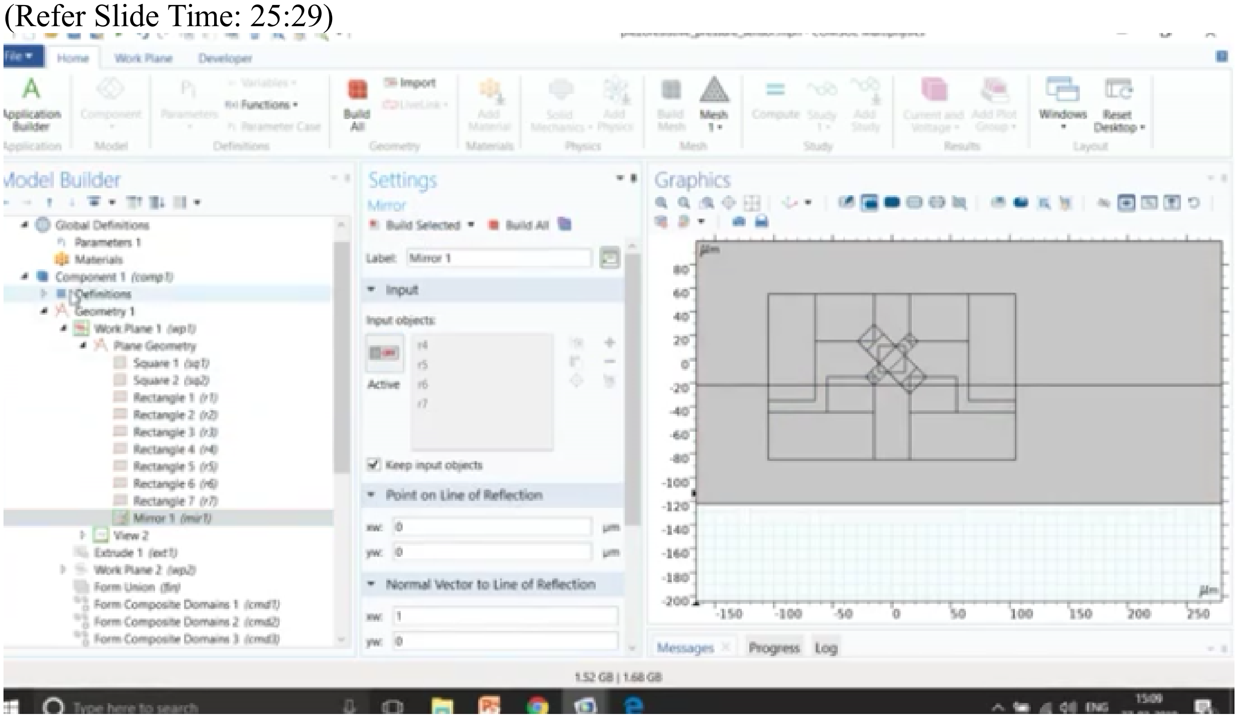


So again the first thing is to make the geometry, so here you will see now little bit complexity in the making of the geometry, again this is not that complex we have used some other domains to simplify the application of this particular domain so while doing the physics, so that's fine, first

thing is to make work plane and then as you can see over here in the work plane we created small, small geometries, so this is the first squared that we made,
(Refer Slide Time: 25:11)

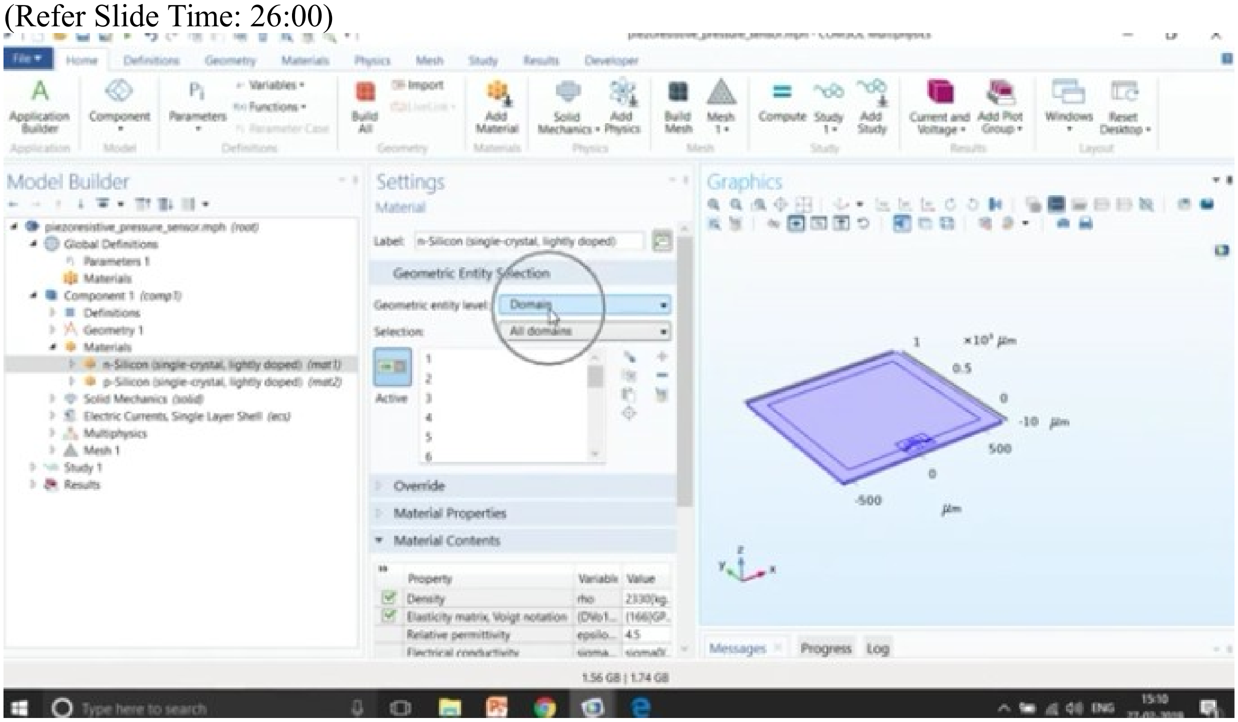


one more square that we made over here, and then slowly and steadily we made the whole structure, if I just zoom in to this part, so slowly and steadily the complete structure was made like this, right.
(Refer Slide Time: 25:29)



And then we extrude this particular structure from top to bottom, okay, and then we do one more work plane, I think this to extrude some, okay, this is to extrude in the other direction, okay, so once you are done with the 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 lightly doped to the complete domain,

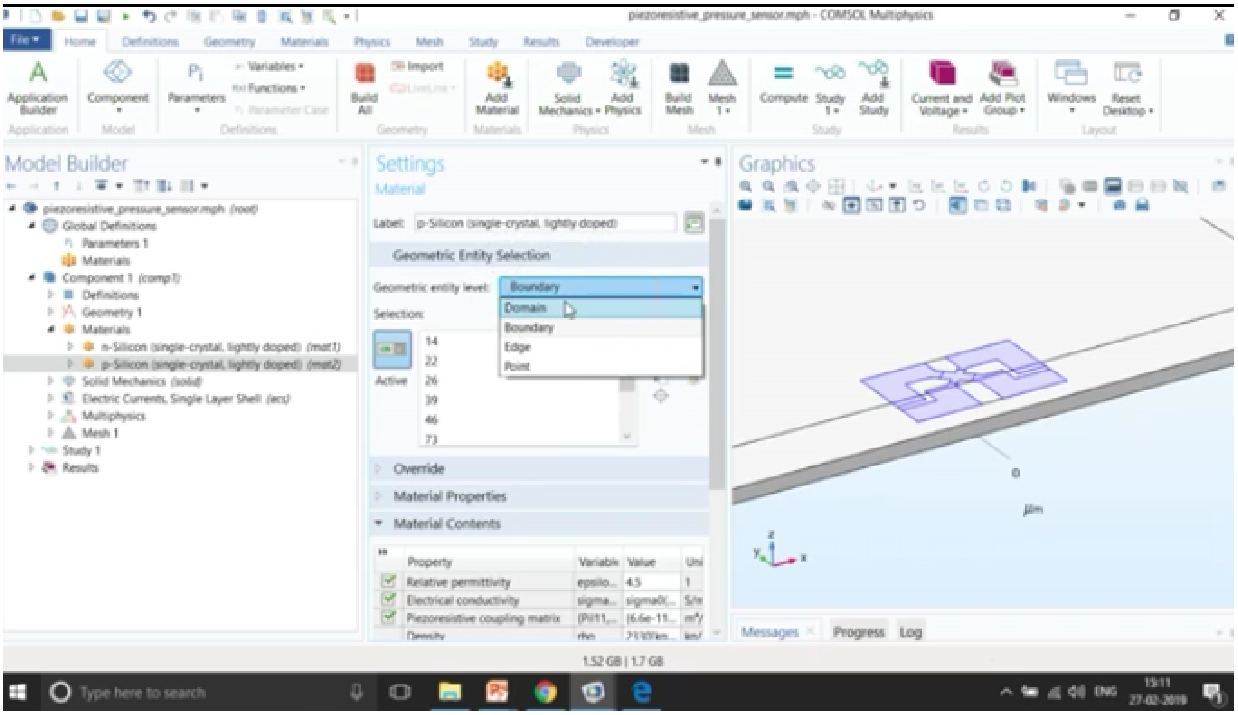
(Refer Slide Time: 26:00)



so this complete domain, so we're again talking about domains, it's very important, what kind of entity level that you are assigning that particular domain, right, so right now the geometry entity level is domain that is volume metric domains in the 3D structures.

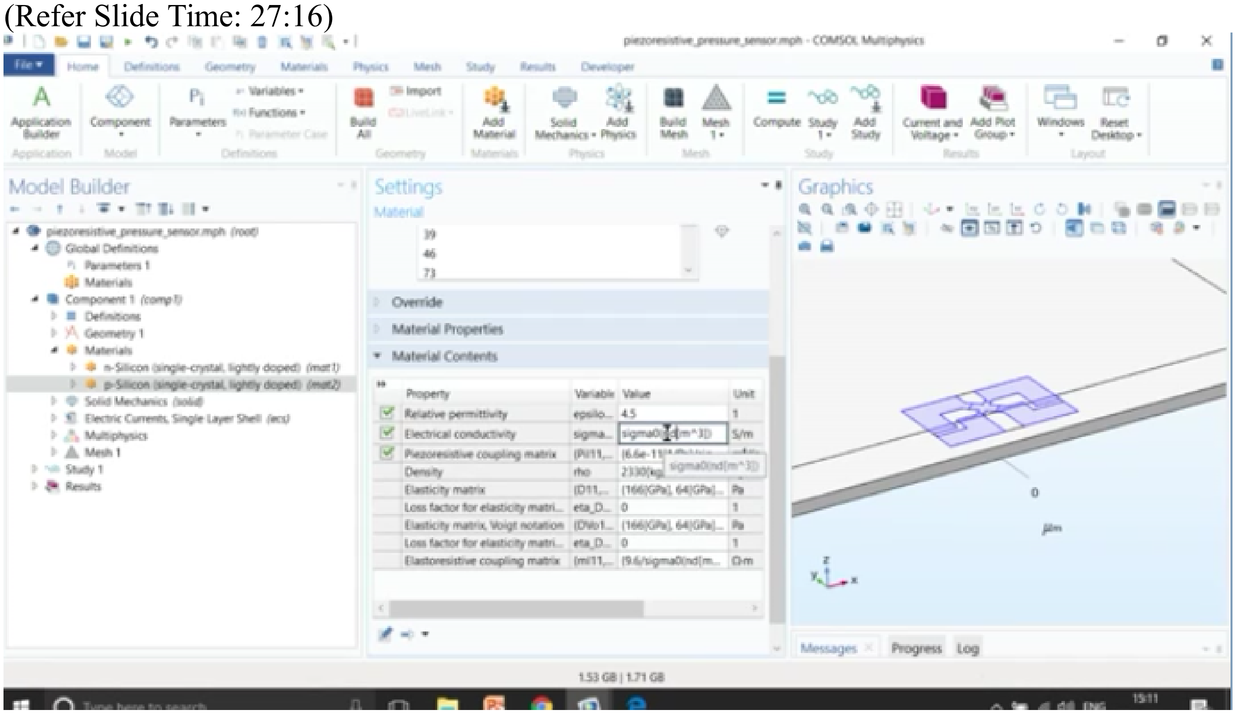
In this case we're assigning scalar value of density and a tensor for the elasticity matrix, again for the P type, now the interesting part comes into picture that we are using boundaries, so we have done away with the modeling, volumetric approaches, but now we are going to model the P type silicon as boundary, this is very important and the result, why we are doing it is the meshing, one of the main reason is the meshing, you don't 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,

(Refer Slide Time: 27:00)



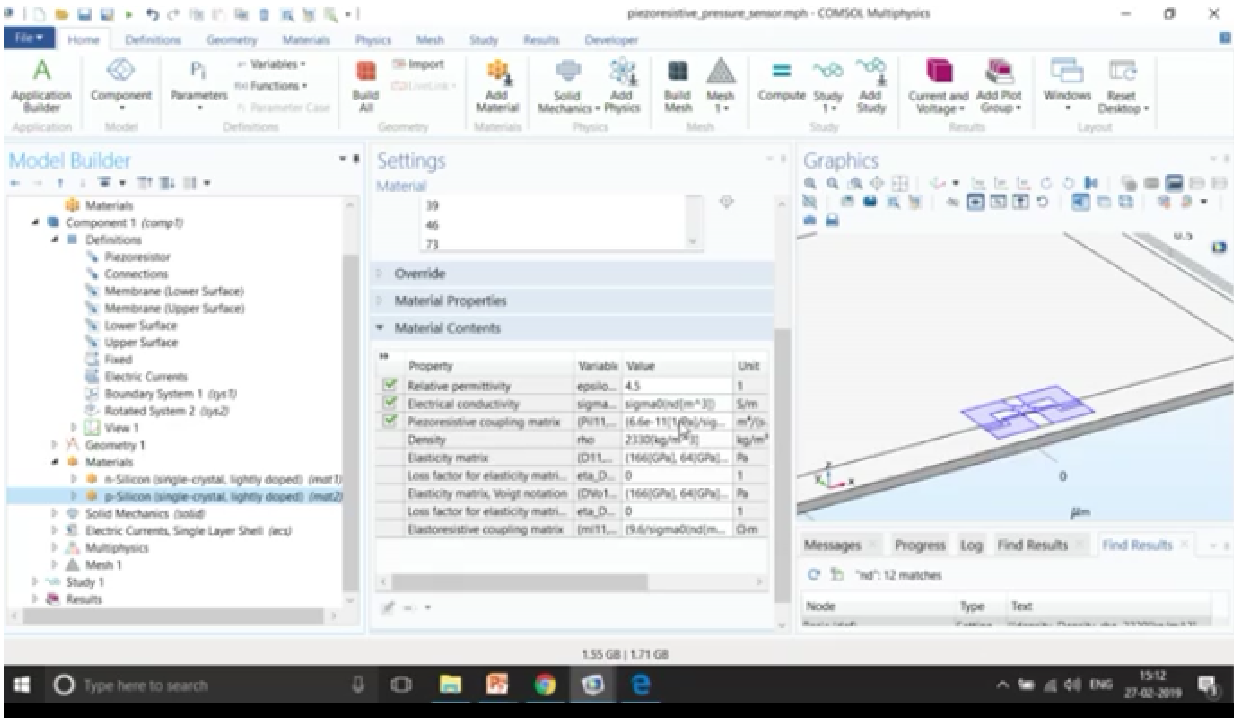
okay, so let it this particular boundaries and then we assign again the connectivity is again a very interesting that the connectivity is a function of ND, so we write sigma naught as a function of ND.

(Refer Slide Time: 27:16)



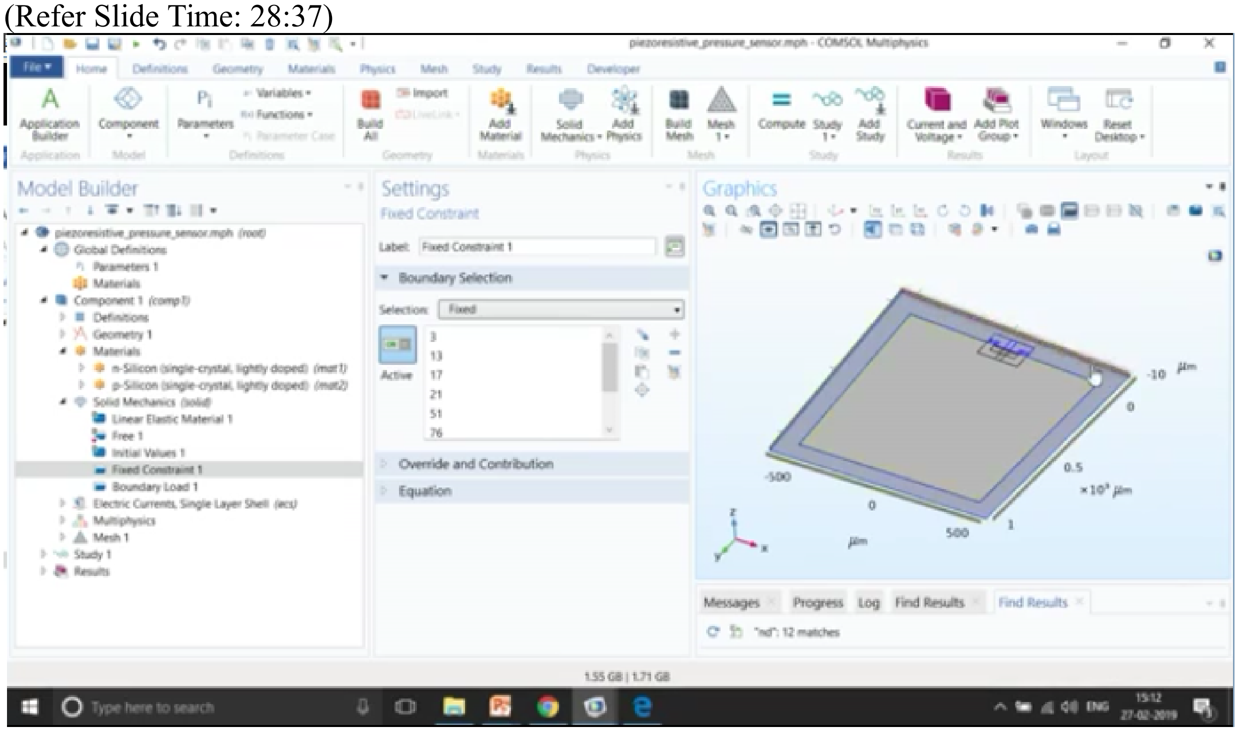
And this ND is the doping, so from where the ND is defined, so let me just go through, it would be defined somewhere else, so if you are not able to find something just right, do a control F and then search for ND, okay, so ND is actually the density of the material that's what understand, okay.

(Refer Slide Time: 28:09)



So once you understand how the materials have been defined, so over here also you have coupling metrics which is a 6 x 6 tensor, 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 fixed constraints on the bottom part, this is the bottom red part that you can see over here, so in the boundaries we give fixed constraint,

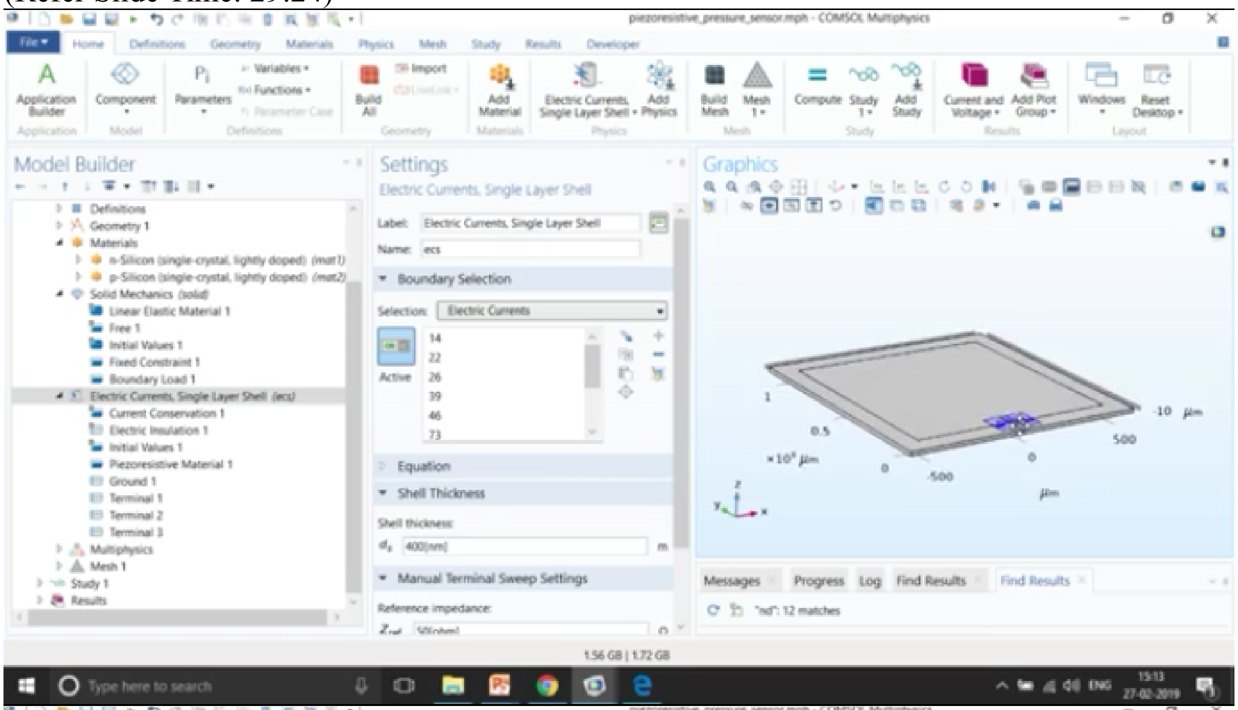
(Refer Slide Time: 28:37)

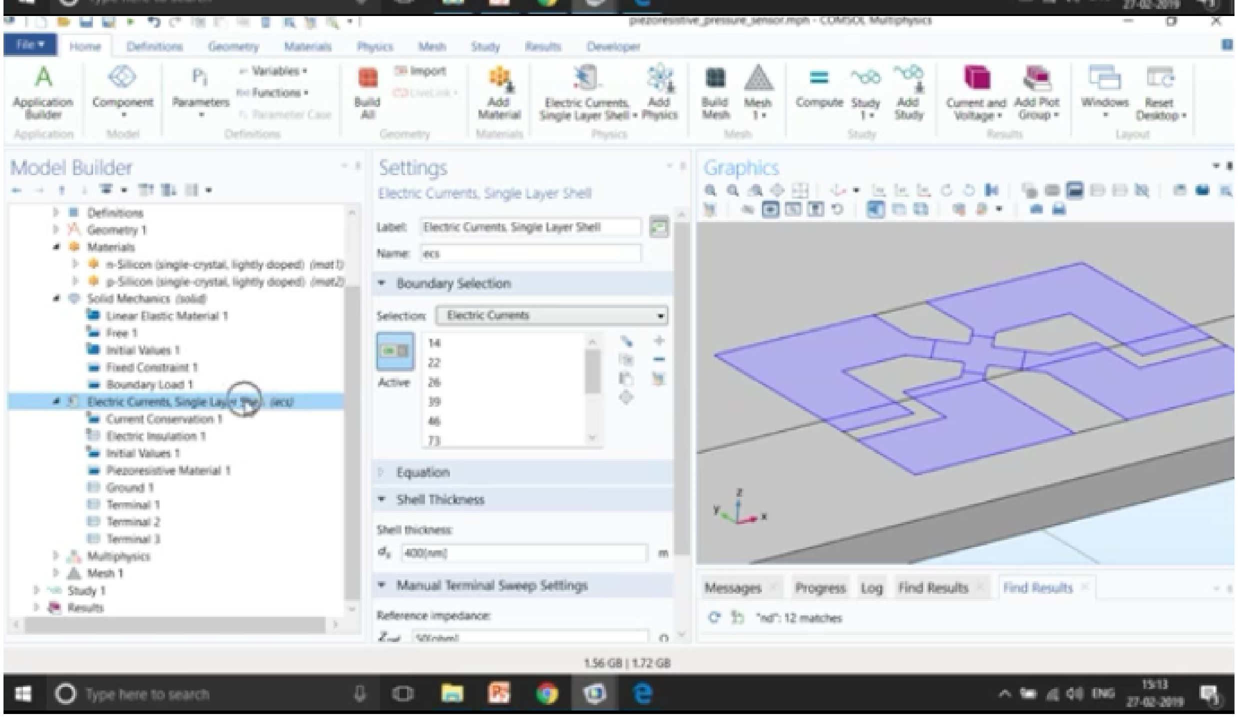


okay, and give a boundary load on the top part, so a force has been applied in the inner surface over here, and what force? We are giving 100 kilopascal of force, right, it's very important of how much force we are going to give, and we are giving a fixed constraint in the bottom, bottom outer edges.

If you have any questions please write it down in the question and answer session, and we will try to respond you through the forum. 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've done away with the volume metric approach and we are only,

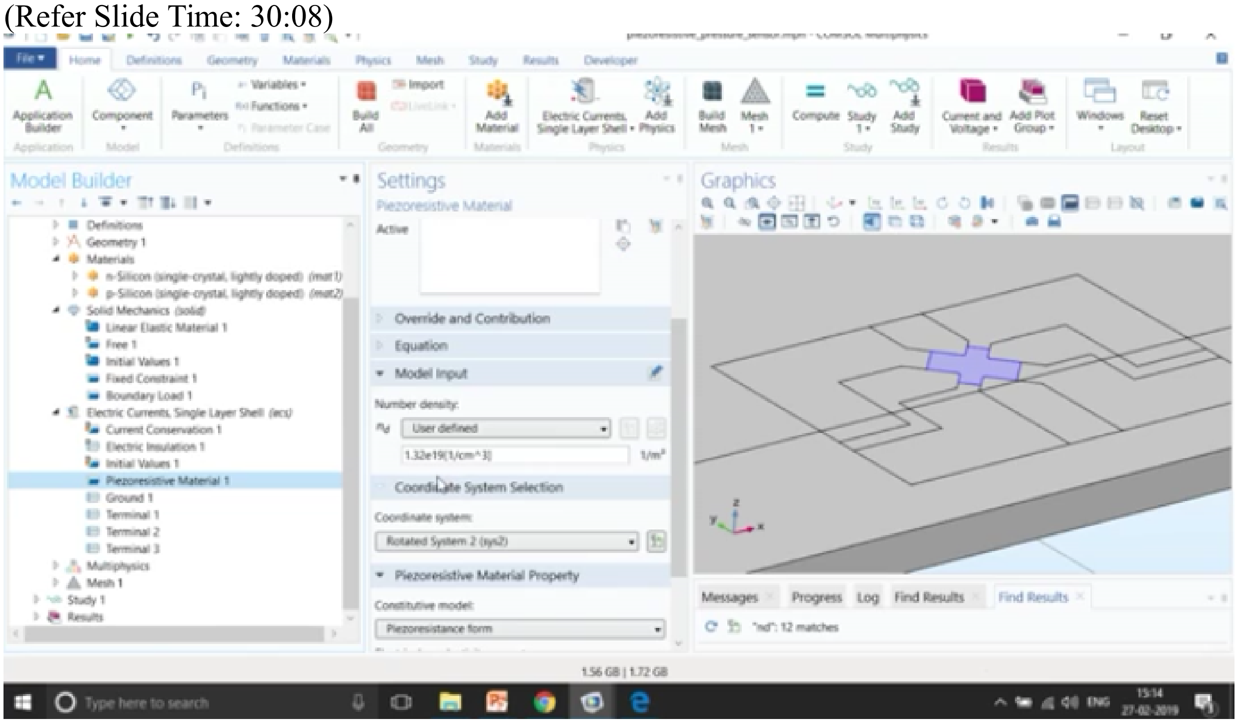
(Refer Slide Time: 29:24)





if I just zoom in we only talking about boundaries, these are all boundaries, so that's why the reason shell has been mentioned over here, right, and we also mention the thickness of the shell, so this is 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's always good, so that's why we use shell layer approach.

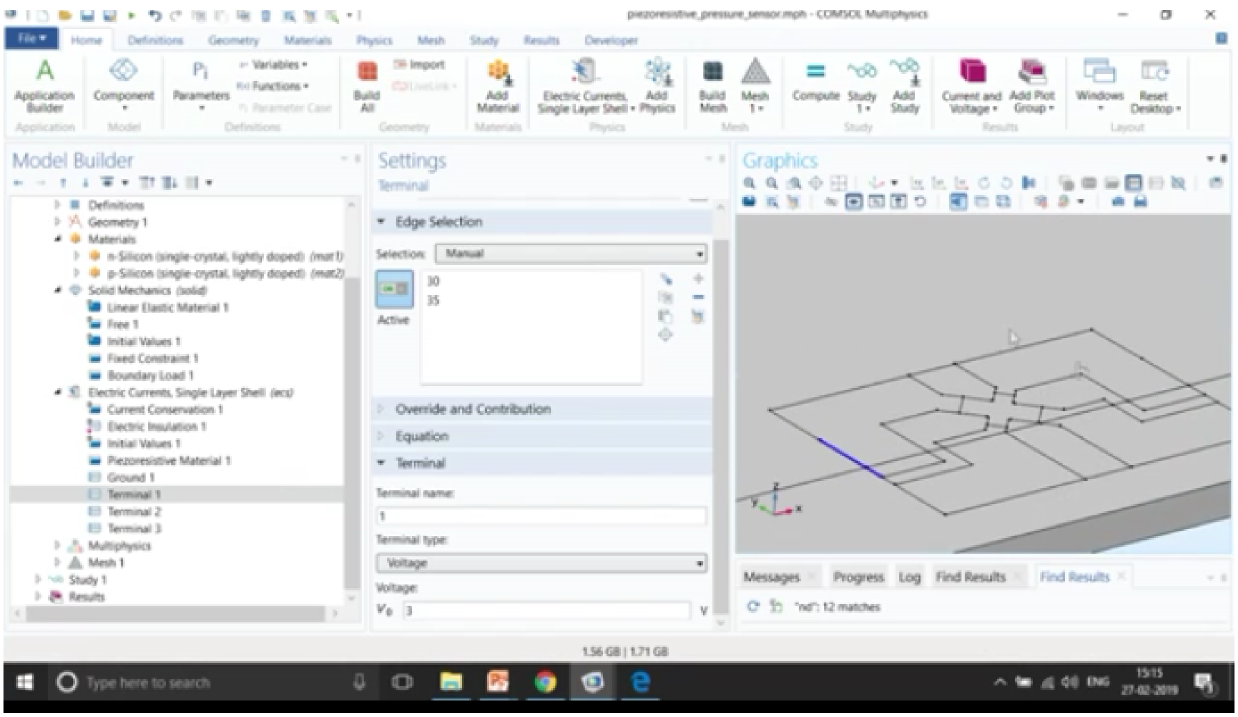
In this case again we use piezoresistive materials that you can see over here, so this is a piezoresistive material, small cross shaped model over here,
(Refer Slide Time: 30:08)



and over here is the number density which is defined that I was searching before, alright.

So the number density is defined over here, and the conductivity, the conductivity is actually a function of ND that is the number density, right, okay. And then again it's using rotated system, so again it's not using the default coordinate system, but it's using the rotated system, so it's very important to know that.

A rotated system has been defined over here, so what kind of angle cut has been given in this particular case, so -45 degree cut has been given over here, okay, 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 part of piezoresistive materials only, so terminal boundary condition have been given over here,
(Refer Slide Time: 31:16)

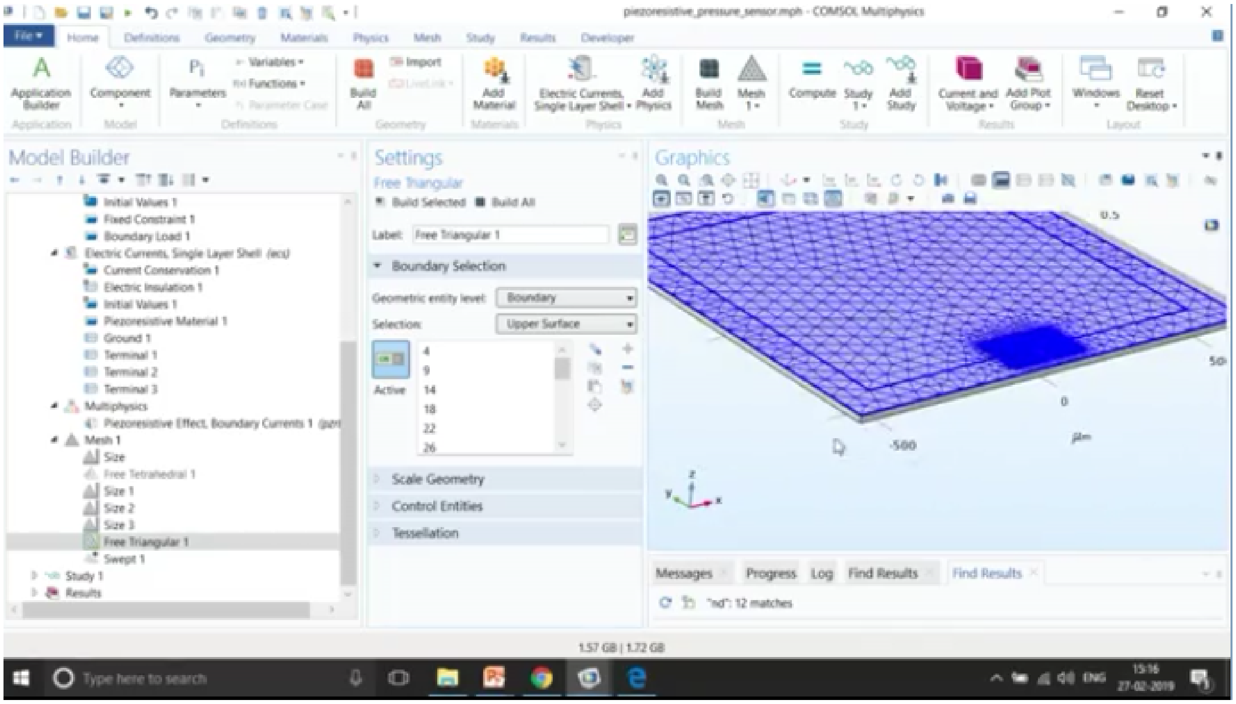


and a ground has been given on the opposite site, so the current is going to flow like this to this domain to the piezoresistor and then finally it will arrive over here.

The next terminal 2 has been given 0 current, and terminal 3 is also given 0 current, this is to reduce the error while we are taking the measurements, okay.

So next part is again the multiphysics part that couples the structure mechanics with the electric currents interface and then finally the mesh part, this mesh part is always interesting, because there are many different ways or approaches to match this particular domain, the way it has been modeled over here is they have given different, different meshing elements to each of the domain, so you can see the piezoresistive material is having maximum element sizes 2 micrometers, other elements of the piezoresistive material have been given with 6 micrometers, and then the edges also have been model over here with a different element size, and then the free triangular top boundary has been modeled, so not the complete volume,

(Refer Slide Time: 32:28)

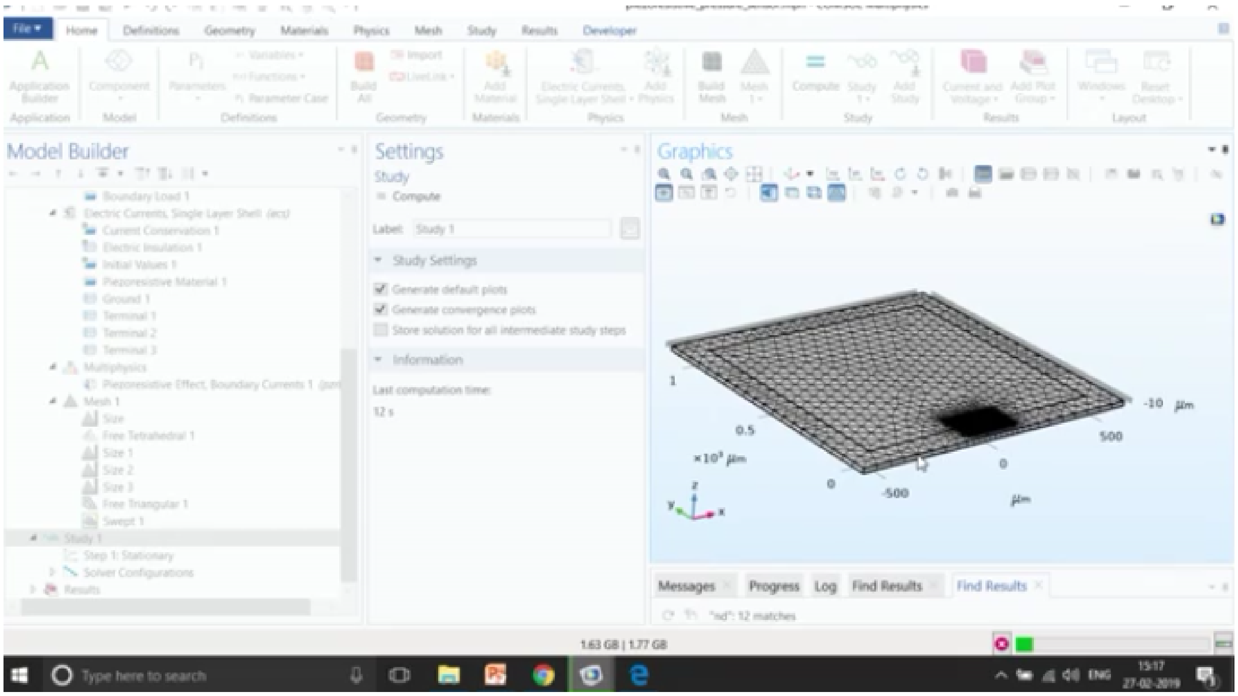


but only the top boundary has been modeled over here, okay.

And then we were sweeping it from top to bottom, right, so right now there is only one sweep, there is one domain with which we need to sweep, if you want to give it maybe for you know 5 domains that you want to see from the top to bottom sweep, so you can just write as number of elements as 5, okay.

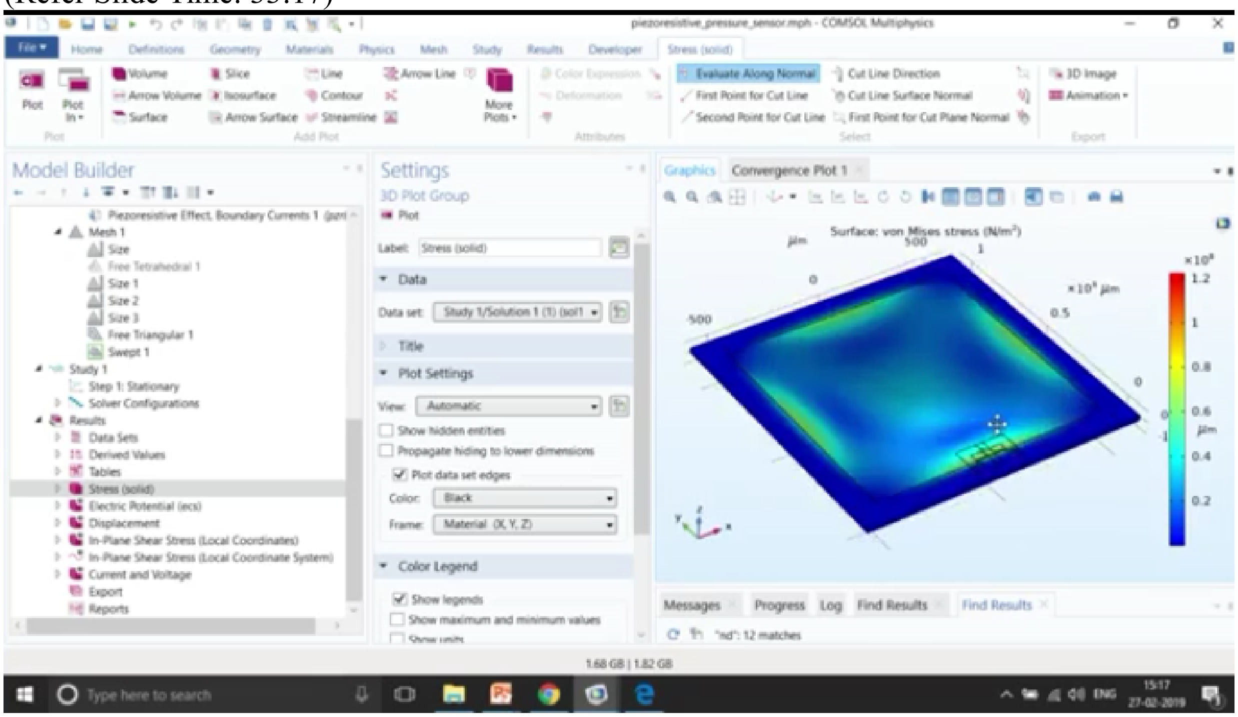
And then we go to study and then run the simulation, right, so let me just delete it and keep whatever is by default, so that simulation results could be perform very quickly, so as it says like around 12 seconds it will take,

(Refer Slide Time: 33:09)



so it’s not that much of time, right, so I can just go and run the simulation.

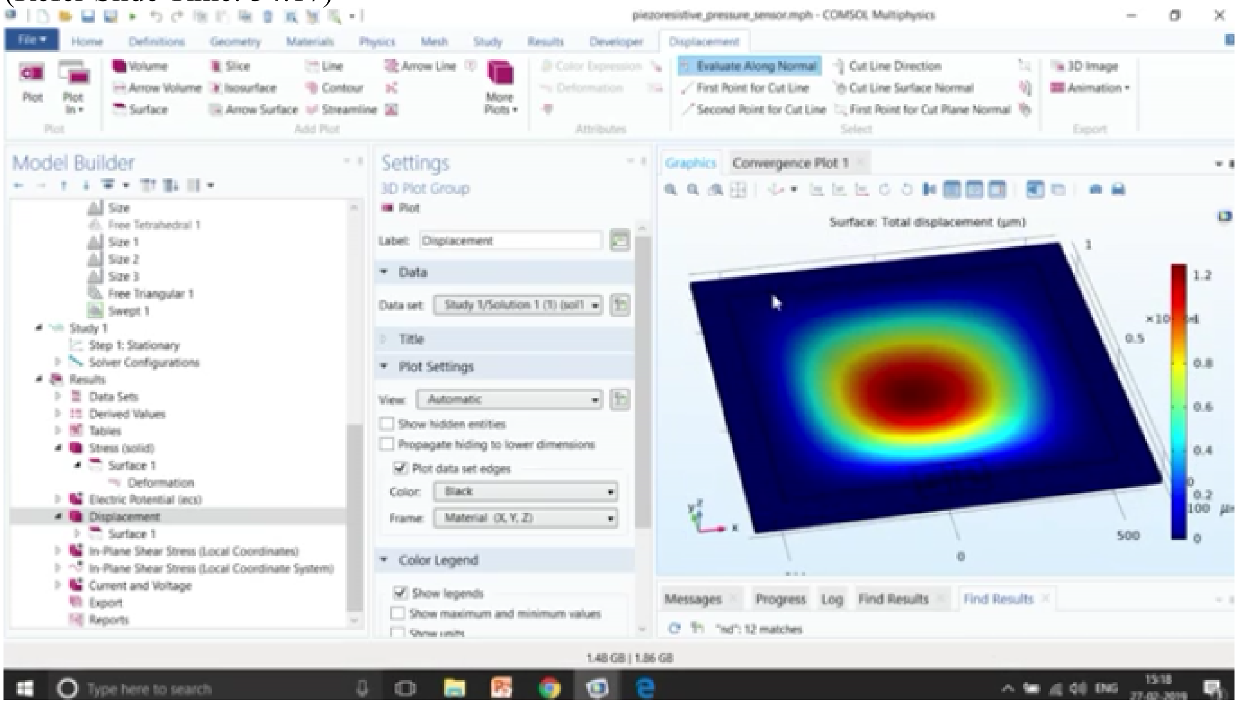
(Refer Slide Time: 33:17)



So now you can see the stresses that is being developed on the surface, and along with the deformation, so it says you how the deformation are going to be, and this is a scaled version of deformation, so if you want to see the actual deformation make the scale factor as 1, the deformation that, the 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’s around, maximum is around 120 newton per MM square, okay.

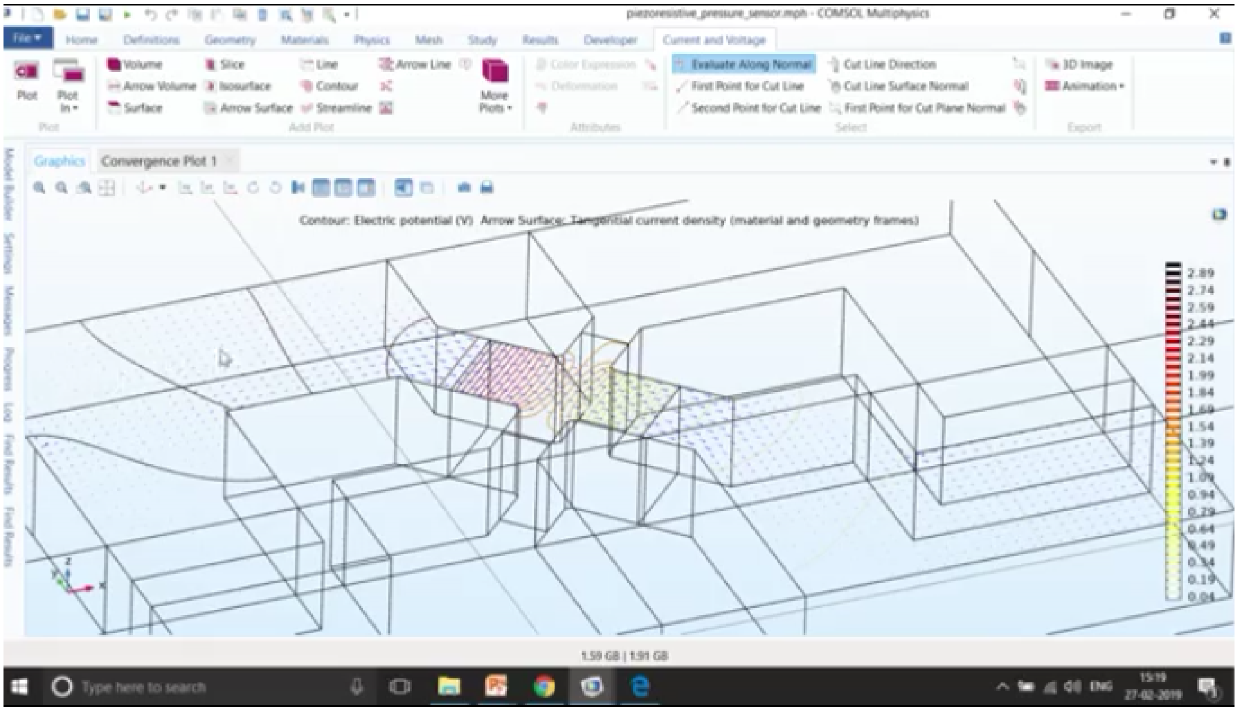
And in addition to it you can also know the factor of safety with which your system will be working, so your revise should not be going more than the factor of safety, so that is also kind of analysis that you can do if your structure is going to fail or not.

(Refer Slide Time: 34:17)



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 and it’s going to pull it down on the center, so we have intend stresses also that you can, you want to see over here, and then finally the currents in the voltages, right, so this is how the currents are going to move, just maximize it, so you can see that the voltage that I have applied is over here, somewhere over here and the colour represents the upper volt, sorry,

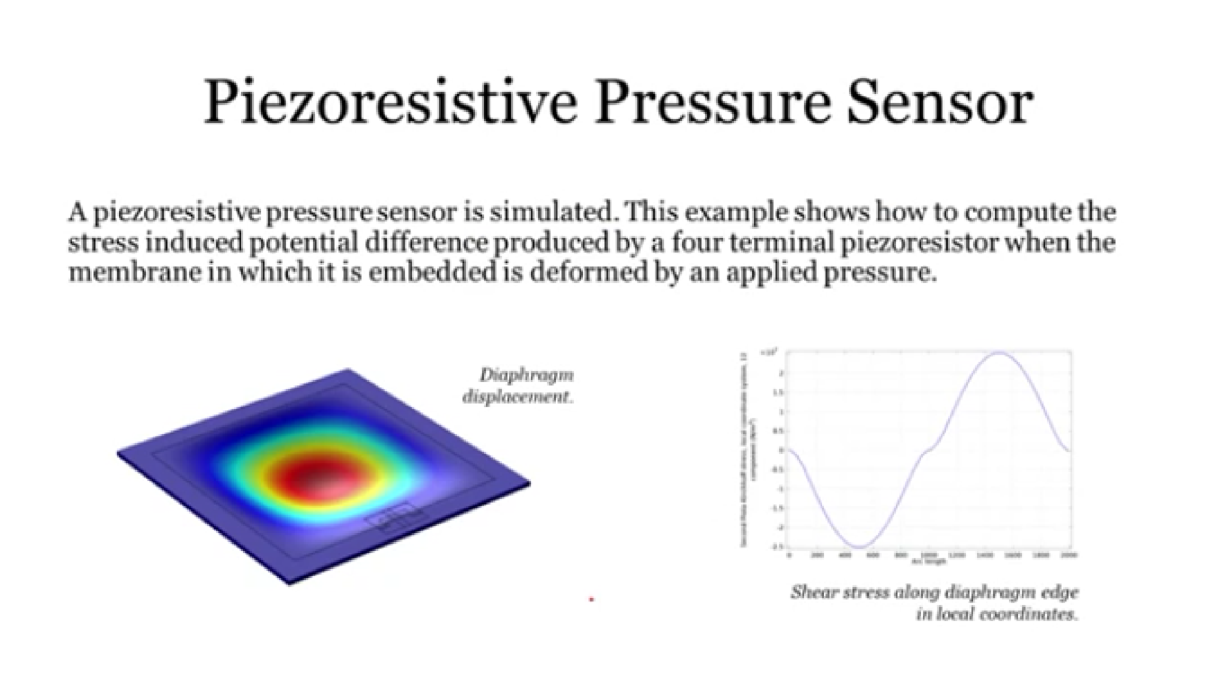
(Refer Slide Time: 34:45)

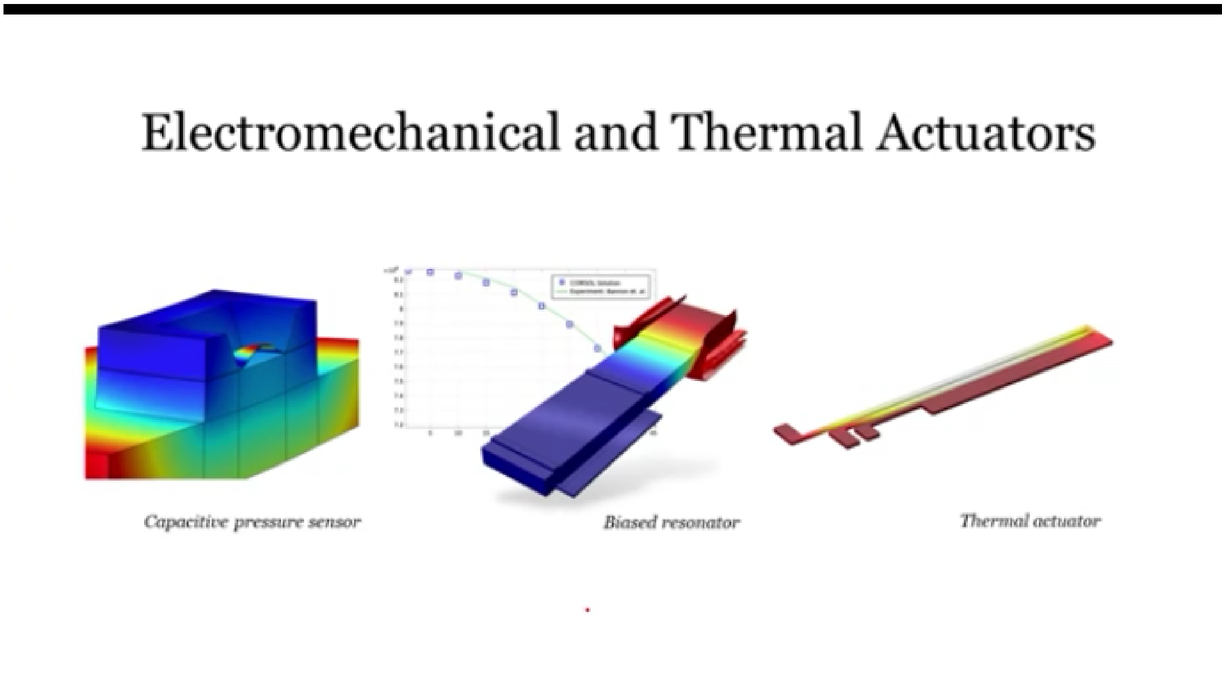


the colour of this counter plots represent the voltage that you can see over here and the arrow plot actually represents the flow of the current from the voltage part over here to the ground over here, this is the flow of current that you can see, right.

So you can actually first make this particular design with COMSOL and then actually you 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 a 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 or all the terminals that you can do it over here, okay, so this is how the approach to model piezoresistive models are, okay,
(Refer Slide Time: 36:04)





so let us go to the next session is electromechanical and a thermal actuators.