INDIAN INSTITUTE OF SCIENCE BANGALORE

NPTEL

NPTEL ONLINE CERTIFICATION COURSE

Electronic Modules for Industrial Applications using Op-Amps

Module 8

Lecture-50

With

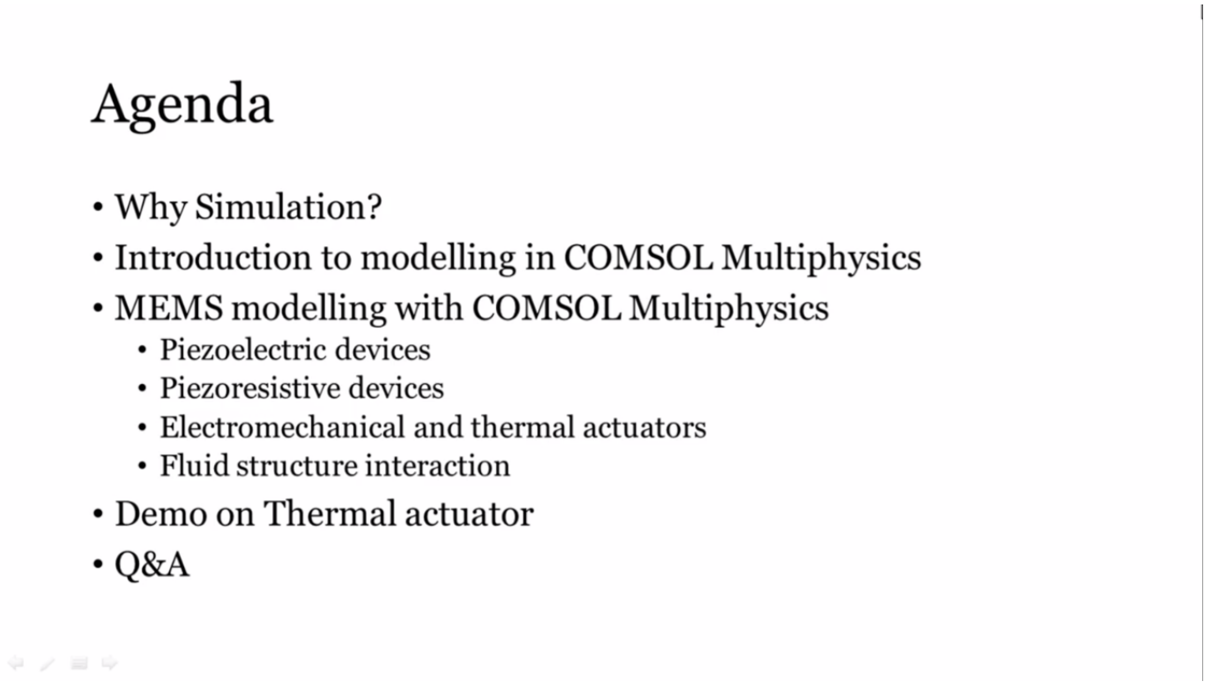
Professor Hardik J. Pandya

Department of Electronic Systems Engineering

Indian Institute of Science Bangalore

Welcome to this session on MEMS simulation with COMSOL Multiphysics. As Dr. Hardik Pandya has already taken a session on how actually you can use the MEMS devices to diagnose, for example, some kind of a cancer, and in addition to that, he also talked about in his other courses that using analogue circuits, you can actually control a particular system or a device. Today, we are going to talk about what is the actual physics that takes place within those devices.

So if you have kind of a MEMS device what actually happens within the MEMS device, what is the physics that happens within the MEMS device. If you want to control your MEMS device in different type of operation how can you do that. So let's begin today's session.

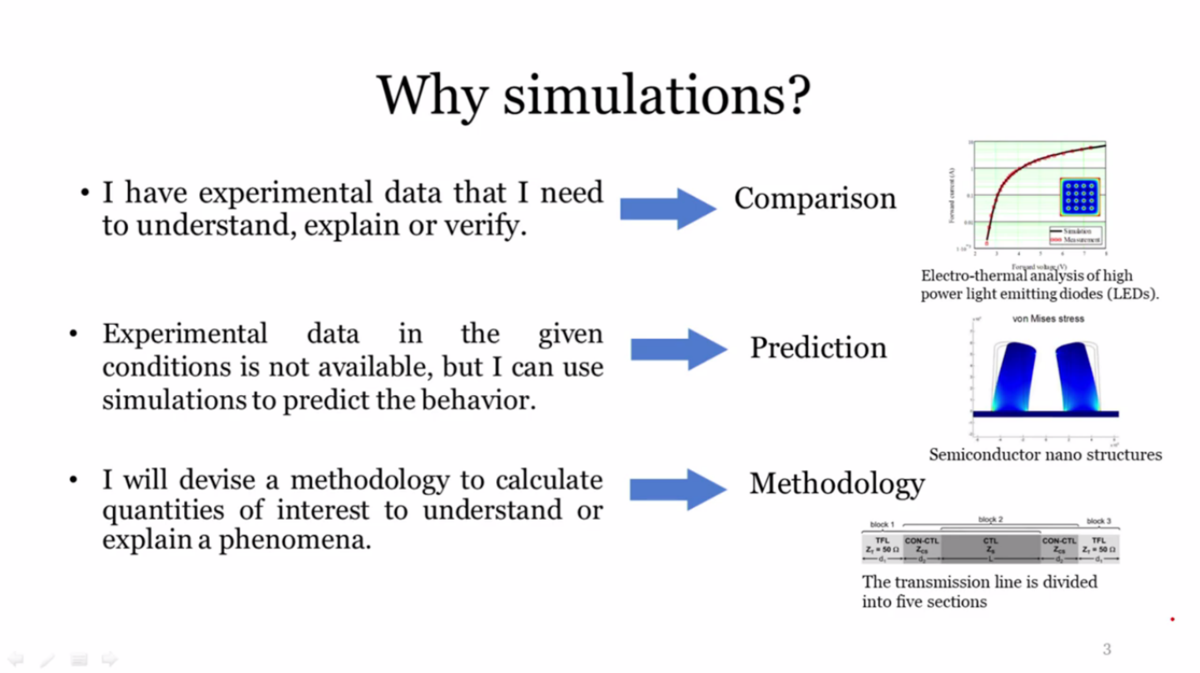


We will start with why simulation, we will try to answer that why people are actually looking for simulation, what are the reasons behind of doing a simulation, why do we need it so much. Then we'll talk about, we'll give a brief introduction of how modelling is possible in a seamless way in COMSOL Multiphysics and then we will talk about different aspects of modelling different devices.

For example, as you can see here, we have Piezoelectric devices, so how can you model Piezoelectric devices, how can you model Piezoresistive devices or if you are interested to model eletromechanical or thermal actuators, how can you do that. And then finally, we will talk about fluid structure interaction.

So as you can see over here, the MEMS device actually requires a Multiphysics analysis. So it's not just the electrical physics that is going to play a role but along with the electrical, because of the thermal heating of your device, how much is the thermal expansion and because of the thermal expansion how much change in the geometry of a structure is. So all those things need to be taken into account and then the electrical physics needs to be taken into the process.

So this is a kind of a fully coupled problem which is going to be solved in today's session and we will tell you how to actually proceed with that. Finally, we will talk about a thermal actuator. We actually take a demo on thermal actuators. So eventually, if you have any questions, you can also pose us the questions and we will try to answer it after the session.



So the first question that comes to the picture is why do we need simulation. So for example, you already have an experimental data with you and you want to actually understand the physics behind that experimental data. So at that time simulation actually plays a very important role. So simulation will help you to understand what are the physics behind those results. If you are looking for some other results, so for example, you are looking for a graph but if it's not exactly this graph that you are looking for, there is something else. Maybe you want the device to saturate it further very early as compared to this graph.

So then in simulation you can actually change the input data, you can change the material properties, you can change the geometry of your device, and try out many numbers of simulation and from there, you will get many multiple graphs and that's what we will see in the demo how to get different graphs for different input properties. From there, we can actually choose an optimized solution of making your device.

So for example, if you want to fabricate a MEMS device, you all know how costly is it to fabricate a MEMS device, but if you use a simulation tool to come up an optimized design then you have some basis on which you can say that I can go ahead and fabricate my device.

Without a simulation tool it becomes very challenging because if you fabricate a particular device and it doesn't work then the complete money that you've invested in fabricating the device is actually gone. So that's why a simulation plays a very important role before you go ahead and fabricate your optimized design.

The next part is to predict. More or less what happens is that there are some of the cases where you do not know how does your system is going to work in different input conditions or it could be a worst case scenario that you want to try out. You are not going to try the worst case scenario on your fabricated device; you are going to use a simulation tool to understand what would be the prediction, what is the maximum displacement.

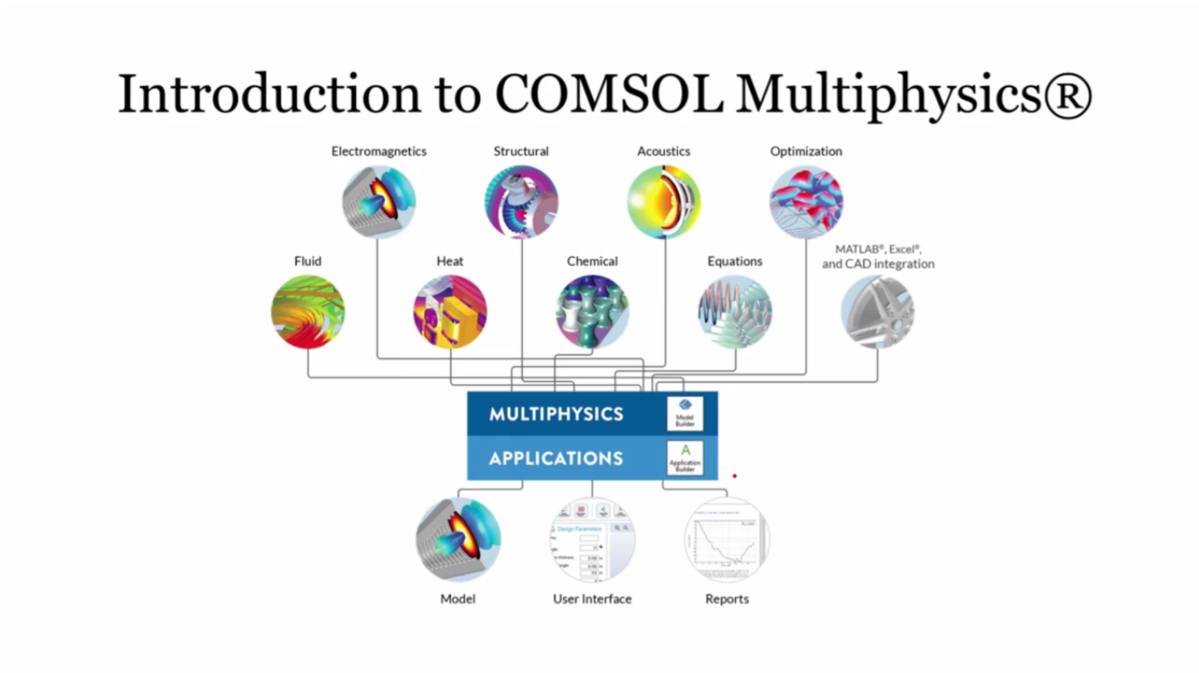
As you can see over here these are two silicon nanostructures and they are kind of – we are given a particular load to this and it's going to deform at a particular angle. So we want to know how much angle maximum that it can go before it is going to crack. So in this kind of problem simulation plays a very important role.

The last part is to come up with a methodology. For example, you already know your physics you already understand the experimental results, you have all the data, but at the end of the day, you need to come up with a mathematical model.

The way to come up with a mathematical model is to do a lot of mathematics. So you need to write your own equations and you need to see if those equations are actually matching up with the actual experimental data. So COMSOL helps you to come up with this kind of mathematical model.

So COMSOL per se is a software which allows you to write your own equations in addition to actually modify the in-built equation.

So if you are solving for Maxwell's equation, for example, you can actually modify a small part of the equation to start with and then see how the results are going to change. So you can actually perform a methodology to verify your analytical results with you experimental results.



Just to give a brief introduction of COMSOL Multiphysics, in COMSOL Multiphysics if you see the world near yourself, you will find that there are no applications or systems where only single physics is in application. So there are two or more physics that are going into which is actually in application. For example, you talk about speaker phones. So you give a particular current to your magnet coils and those actually vibrate the diaphragm.

So here the electrical physics is going to happen when you pump in the current. It's going to create the magnetic field. Then it's going to create the forces that is the Lorentz force that is F=qvxB and that is going to make the diaphragm vibrate.

So this is a complete Multiphysics simulation that you can model in COMSOL. So we already have many other model files, for example, the speaker example that I told, we already have those model files.

In addition to that, if you want to solve a single physics at a time, you can solve it using electromagnetic module. You have structural module where you can solve the deformations. Then you want to model acoustics, you can also model that. Then you have fluid, so you want to model turbulent flow, lamina flow you can do that.

In heat transfer module, you can model convection, conduction, and radiation. And then finally, we have chemical reaction module where we can model electro-deposition, battery and fuel cells, electrochemistry all that can be modelled in chemical reaction modelling.

In addition to it, you can actually write your own equations, let it be in the form of general form or coefficient equation which is a very simple way to write your own equations. So at the end of the day, COMSOL is nothing but solves the pre-built PD equations. There are a PD equation to electromagnetics, there are a PD equation for structural.

So if you talk about Multiphysics coupling then it becomes a coupled PD equations that you are solving. So eventually you get a transfer matrix and then taking inverse of it, you get the actually solutions. So whatever you do it is using the model builder that you can see over here and then finally you can create a very simple app out of your model.

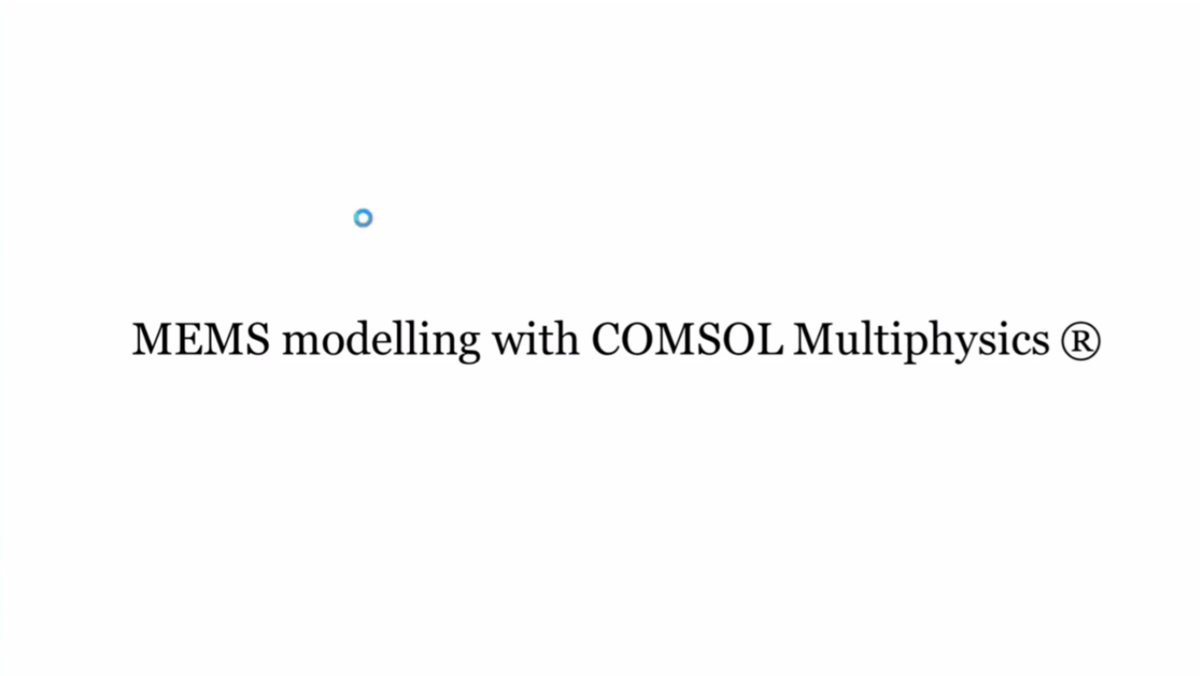
So for example, you are an expert. So the people who would be attending this session, could be a faculty, it could be a student. You could be also a service engineer, design engineer and R&D engineer. So for all of them, what happens is that, let me take an example of an R&D engineer. So he is a very intelligent or he has a very specific knowledge in electromagnetics, for example.

So in electromagnetics he has lot of experience. So he can actually build a model and then make a very simplified interface of that model in the form of app and that app he can share it with his other service engineers who are having a good knowledge in heat transfer of structure.

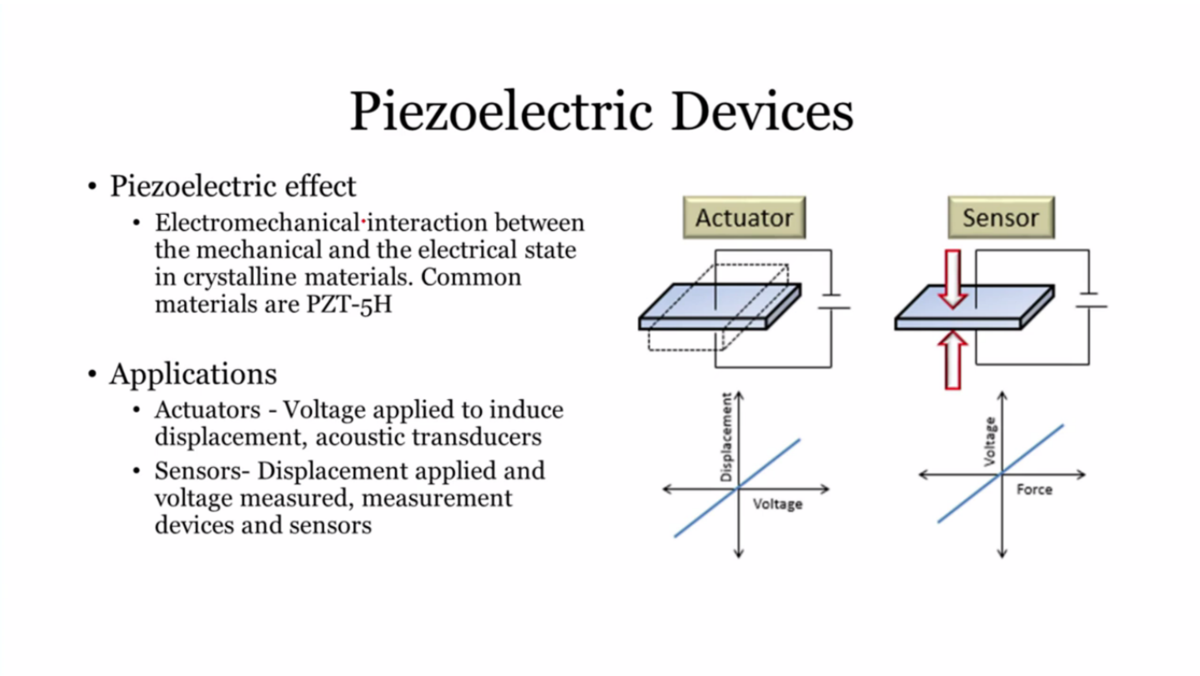
In that simplified app, they can actually change the sum of the parameters and get the results. For example, if you talk about speaker, some company is creating a new design for a speaker in a mobile phone. So for the electromagnetics part, R&D engineer can actually work and create an app out of it. The other person from acoustic field he can share it with acoustic person and then he just need to change the results and then see, what is the pressure level across the room surrounding the speaker.

So such kind of simple interface are known as app. So in the demo also we are going to show that not only R&D engineer but in addition to it, you have faculties who can share these apps with the students and they can actually change the parameters and they can get different, different results out of it.

At the same time, a student can also make an app, a very simplified app and then share it during a presentation and they can see if the professor asks them to change some kind of parameter, they can actually change it and then show the results at that time itself.



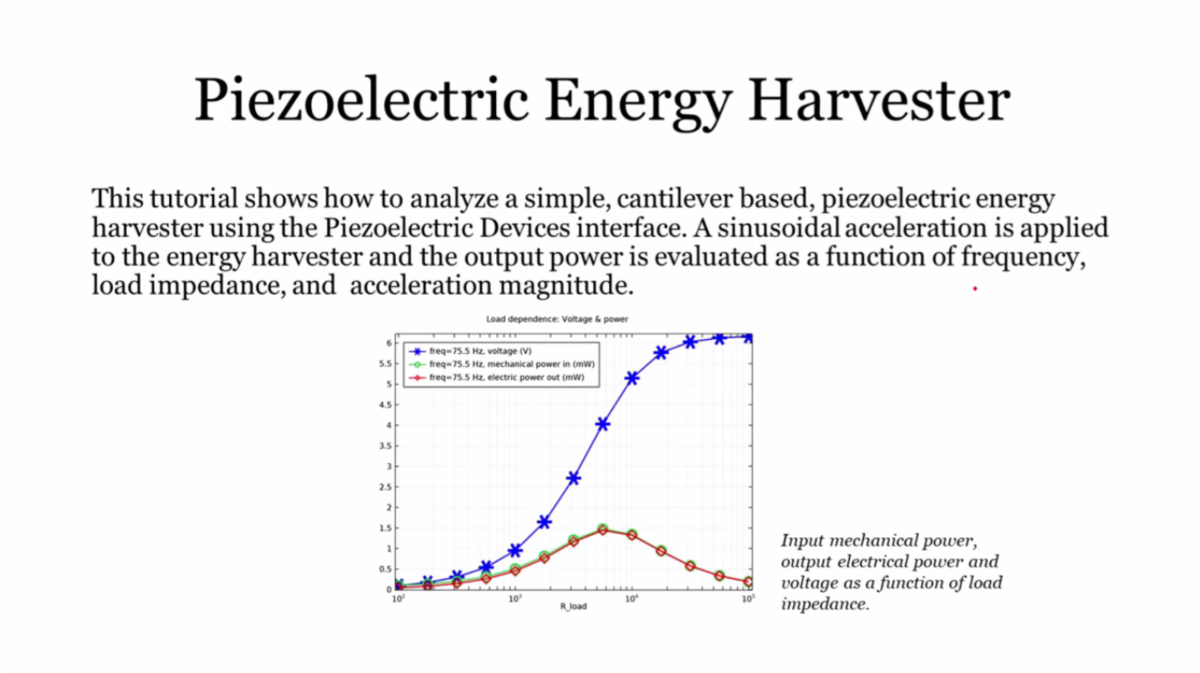
So let us go with the different types of MEMS modelling that is possible with COMSOL Multiphysics.



The first type that comes in today's agenda is Piezoelectric effect. There are basically two types of devices that you want to model. The first one is the actuator type that means you want to have some kind of motion or you want to have some kind of displacement. The input that you have is voltage. So in this particular example, if you give a particular voltage, there would be a particular deformation that is going to take place.

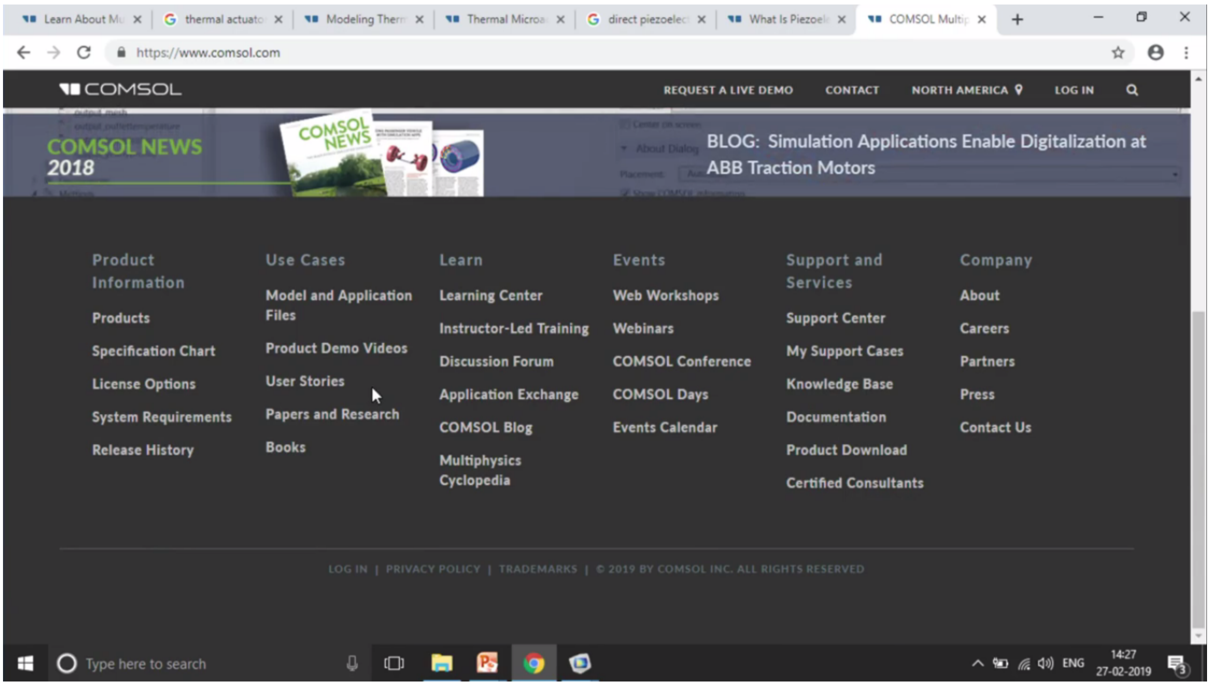
In another kind of application it could be a sensor. For example, you want to sense how much is the displacement of a particular device or you want to sense how much is the stress developed within a particular device. So in that kind of application, you can use the

Piezoelectric device in an inverse way, opposite way in which the force is applied, the force is actually an input and then you get a particular voltage as an output.

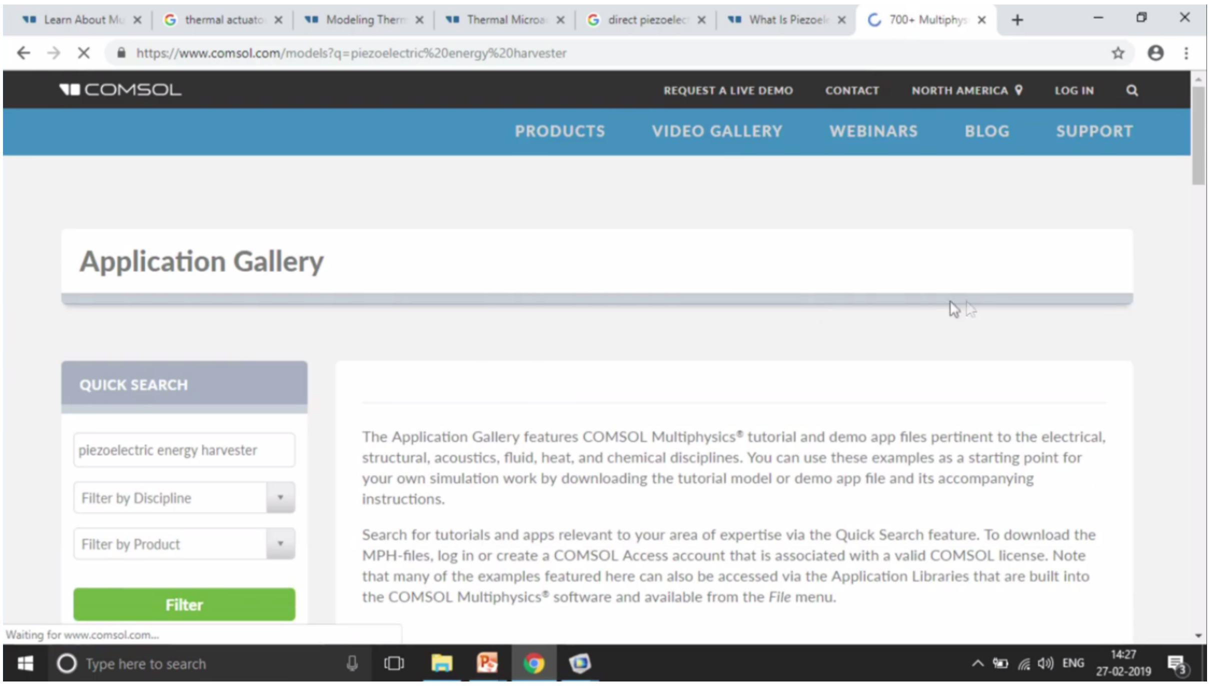


So to begin with one of the example is Piezoelectric Energy Harvester. This is a very interesting example because nowadays people are ahead on how to save energy, recover back the energy that we are wasting in day-to-day life. For example, my friend is actually jogging in the morning. So while he is jogging, his shoes are like vibrating and he wants to actually save those vibrational energy in the form of battery that he can use it for his wrist band, for example.

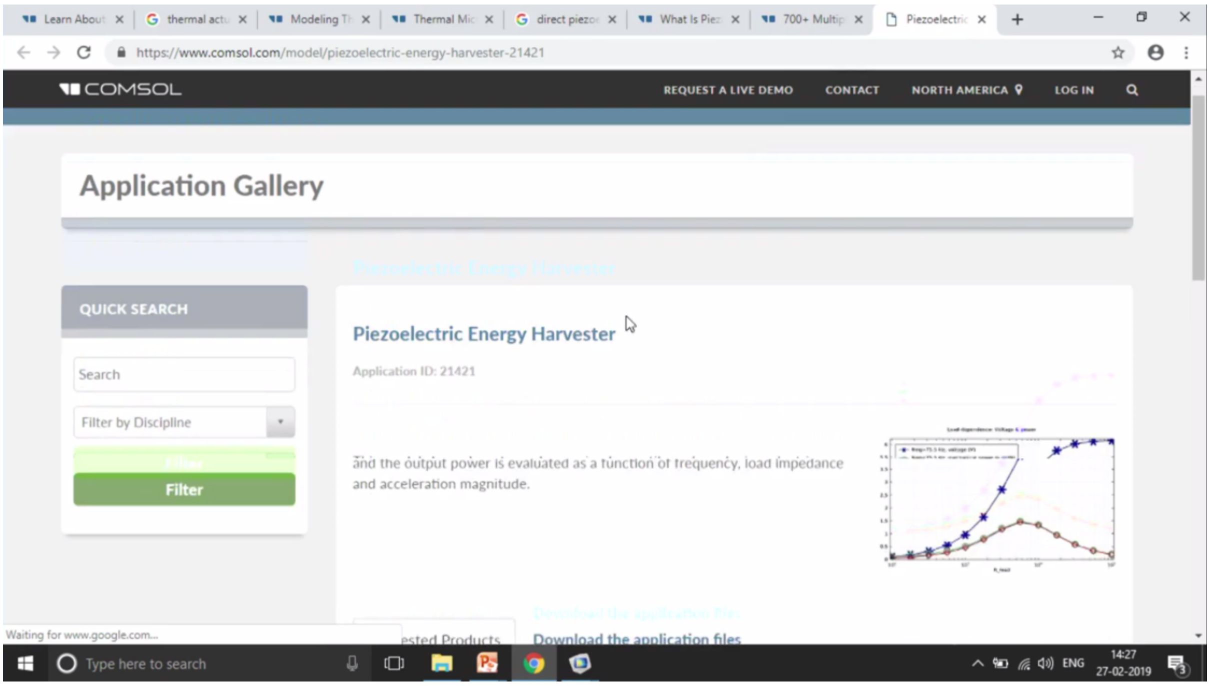
So the way to do that is harnessing the energy of vibration and then converting back into electrical energy. So in this kind of example, having model in this particular example. So we also have the application model for it. For example, if you want to just see this example model you can go to the COMSOL website.



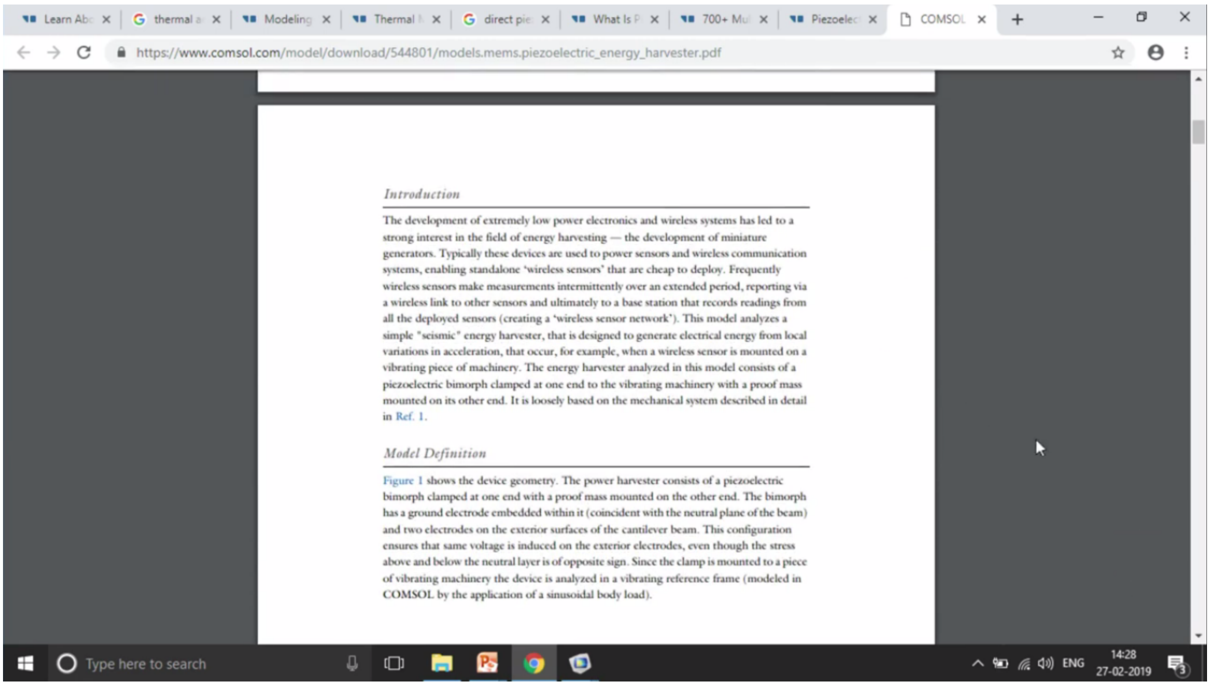
And then you can go to this application, Models and Application Files.



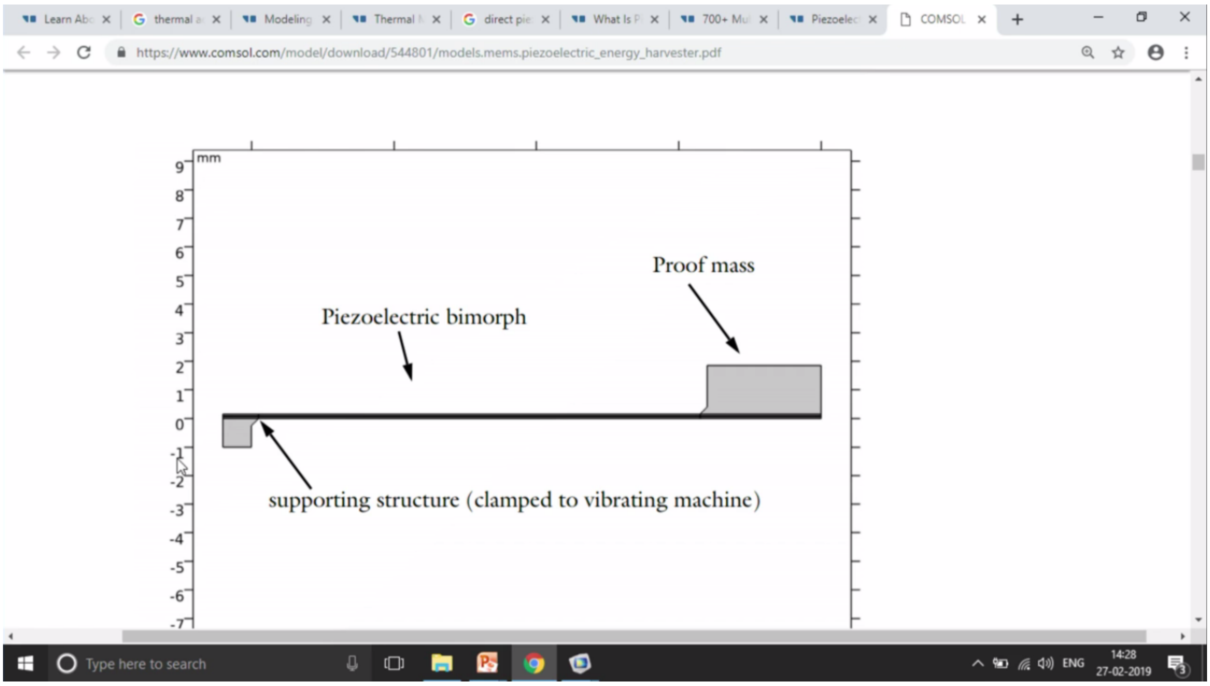
And over here, there are lot of model files over here, and here you can actually search for that particular file. For example, I am looking for Piezoelectric energy harvester, so Piezoelectric energy I am just searching and harvester, let me just search for harvester also.



So you see this is the example model that I am getting. So you can also download the files. You can see, if you have logged in with your account over here, you can also download this MPH file, and you can also see the documentation.

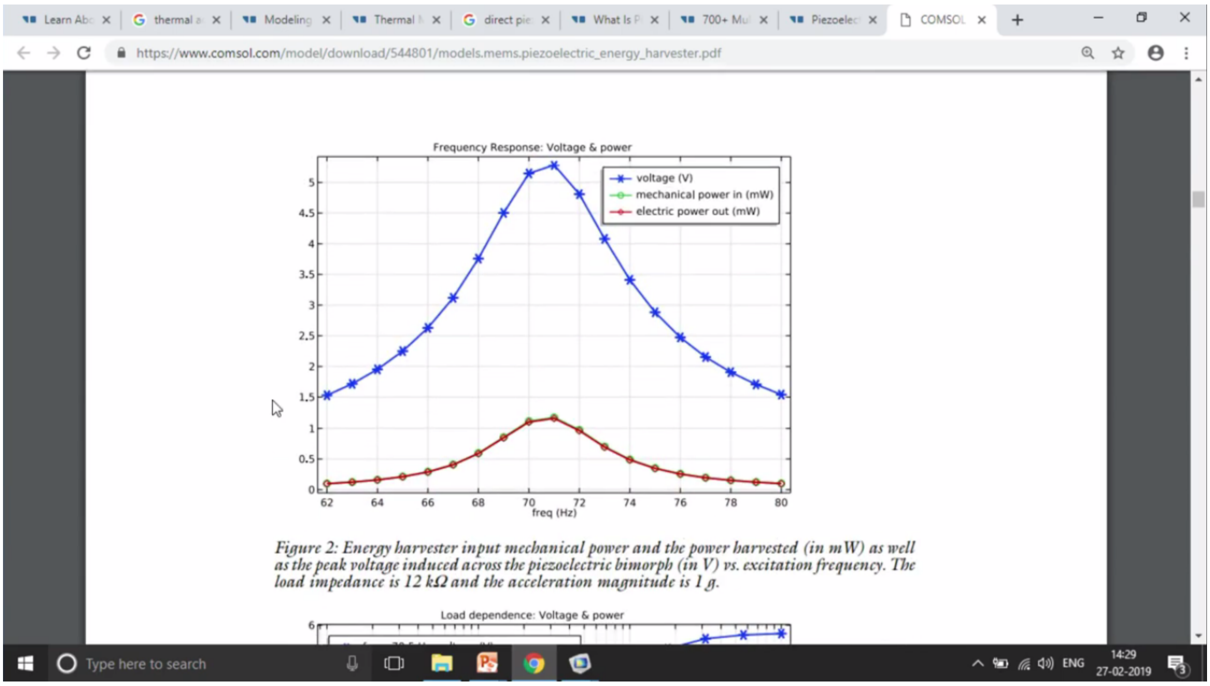


So documentation actually has all the data, all the information. So it first talks about the theory part, then it talks about the setup, the problem setup.

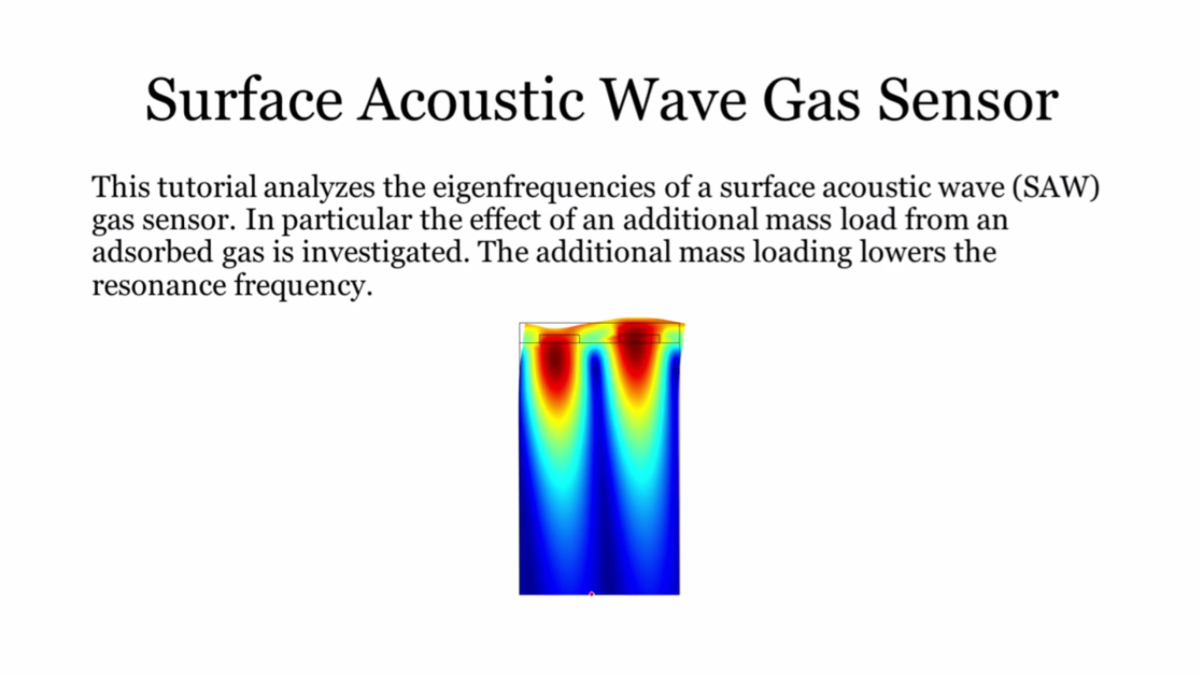


As you can see over here, this is a problem setup where they wanted to understand, they wanted to take the vibrational energy from a motor, for example. So when the motor are working, a lot of vibration is happening. So any vibrating machine would create a lot of stresses on the surface of the machine. So they want to harvest those stresses or the forces, vibrating forces.

So in this case, I just need to maximize it. I am not sure if you can see it but this is a bimorph structure. Bimorph structure it means there are two layers of Piezoelectric which is actually coded with a particular metal strip and then we are going to give a vibrational force from proof mass. And then we want to vibrate it for different frequencies that you can see over here.

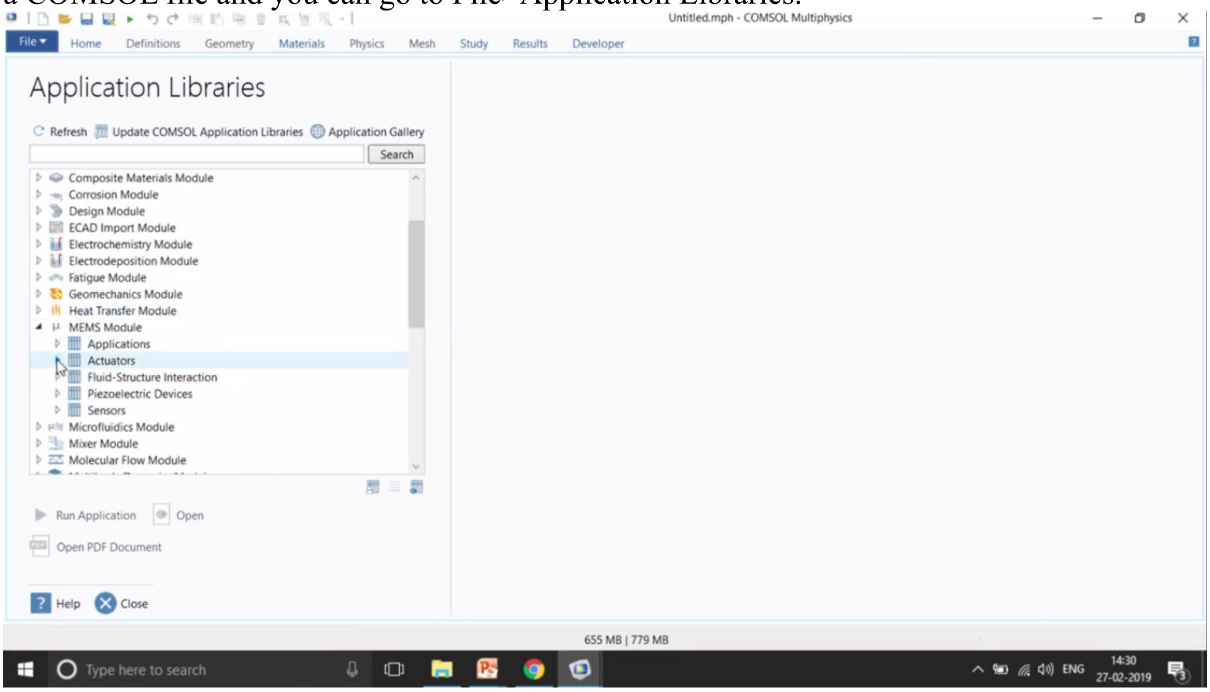


And the peak which is available over here is a resonant frequency for that particular device. So at that time you will have the highest voltage that you can harness back from your device.



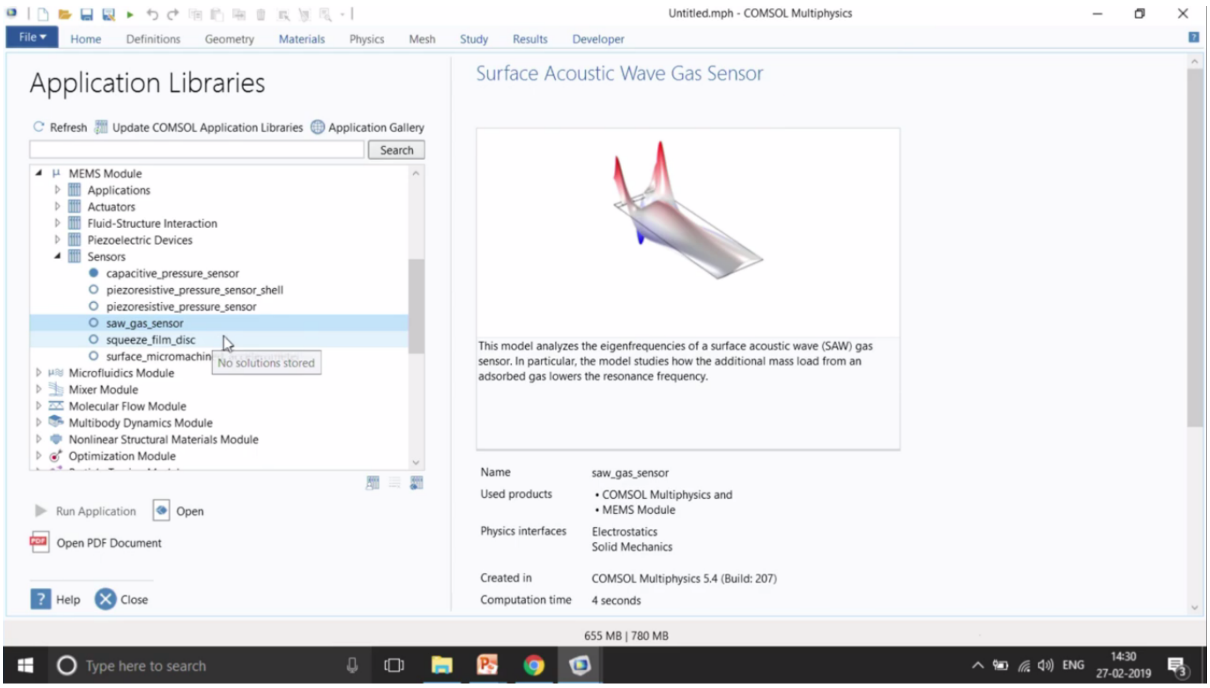
Another example is surface acoustic wave gas sensor. In this particular kind of an example or this particular kind of an application, what happens is we want to understand the composition of a particular gas. We want to know what is the constituents of the gas and also the concentration of the gas. So we actually try to modulate along with a Piezoelectric device. So let me go ahead and open that particular model.

So what I will do I will go to COMSOL. So this is the first thing that you will once you open a COMSOL file and you can go to File>Application Libraries.

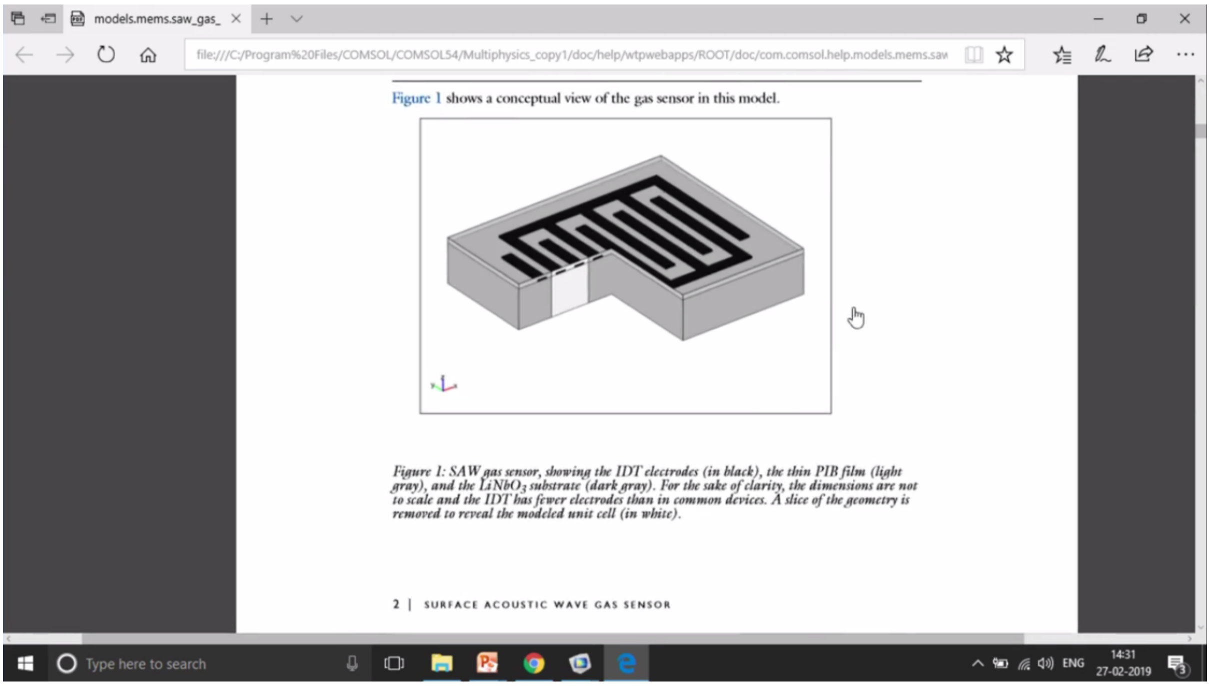


Right now I am going to focus on MEMS device. So you can see there are many modules. You have a AC/DC module, you have acoustic module, battery and fuel cells, CFD, heat transfer. As of now we are going to focus on MEMS module. In this, so in the MEMS module also there are many application models. So you have many actuators that will come at a later stage. We have fluid interaction, we also have Piezoelectric devices.

So right now we are going to talk about surface acoustic wave. So that would be coming in the sensors.



So this is the SAW gas sensors. We also have an example of glucose sensor. So that is also a part of how you can understand the chemical reactions with the help of an electronic device. So this is the example model that I was talking about, SAW gas sensor. You can also open the documentation, try to understand what is the theory behind the gas sensor. So any gas sensors, micro electromechanical structures has an IDT. So this is an actual structure as you can see over here, this is the electrodes, the black structures.



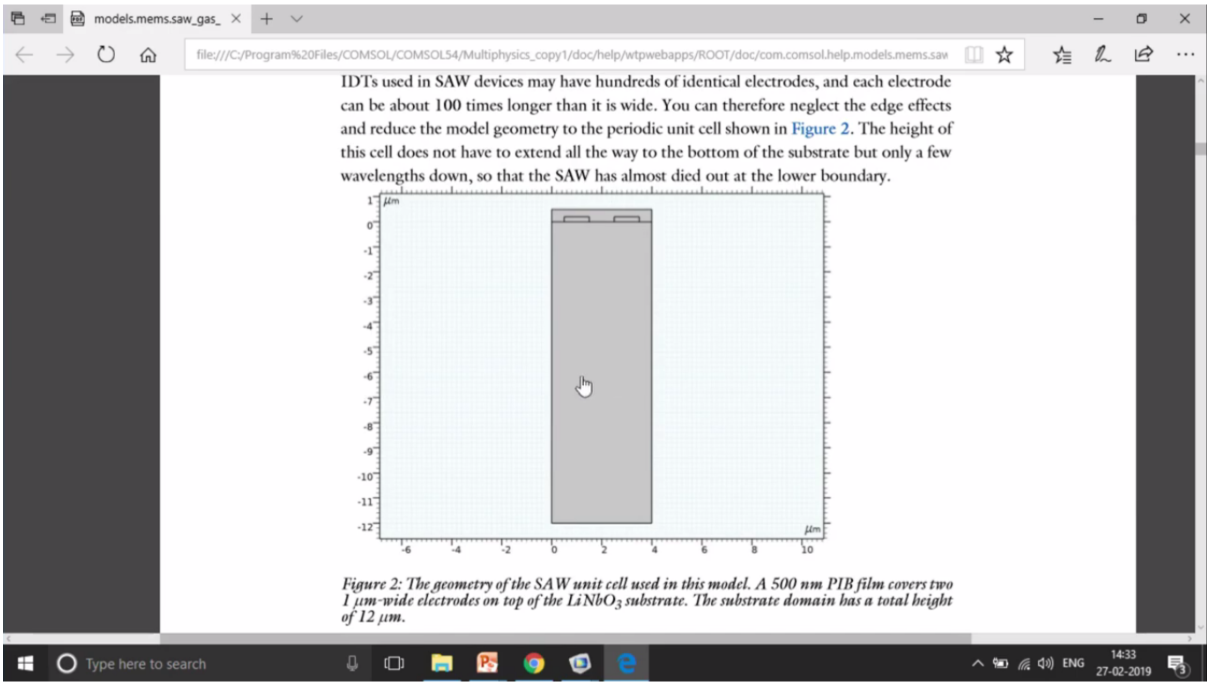
So it's like the fingers grabbed with each other and once the gas actually passes over this or gas has its own connectivity and its own impedance. So if you are able to understand those impedance we can actually back calculated or do kind of a reverse engineering to understand what is the composition of the gas, what is the concentration of the gas, what are the different constituents of the gas.

So this is the actual structure. But if you are a simulation engineer or you are a student or you are a faculty who is trying to model this surface acoustic wave, you will not go for a completed 3D simulation because it will take lot of time. A simulation engineer or a person who is trying to do a SAW gas sensor will come up with a technique with which you can simplify your model.

One way to simplify is to use a 2D cross-section of your model and then give a periodic condition on the edges, for example.

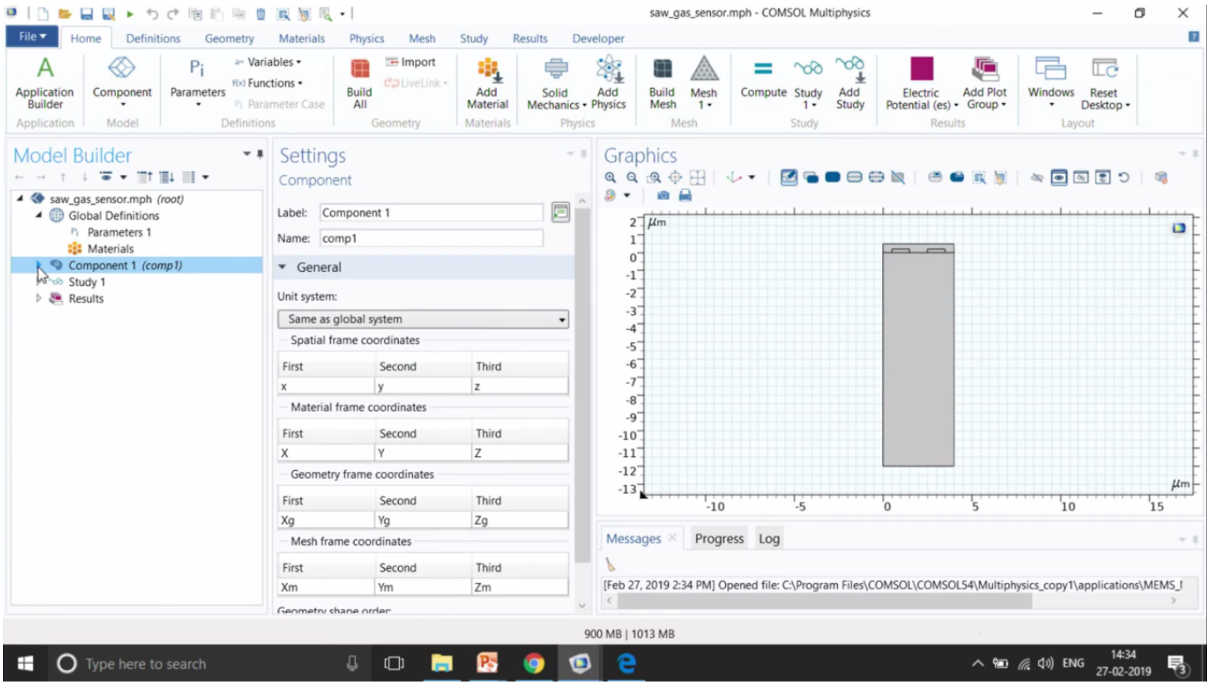
So in this case you can see a bright spot over here. So we have an actual structure, SAW structure, surface acoustic wave gas sensor and it is having IDT at the top and then it has a thin PID film that you can see over here and then it has lithium niobite oxide substitute in the bottom.

So instead of modelling the complete 3D what we can model is only a 2D structure that you can see over here which I am marking from my fingers. So only 2D structure is being modelled over here.



So the 2D structure is now what you can see in your screen. So this is actually what is modelled in this case, not the complete 3D geometry. This is the most simplified and this is what we should always see or we should always start with this kind of simulation which is very simple but it captures the physics which is more important.

So let me just go into a SAW gas sensor and open that particular model.



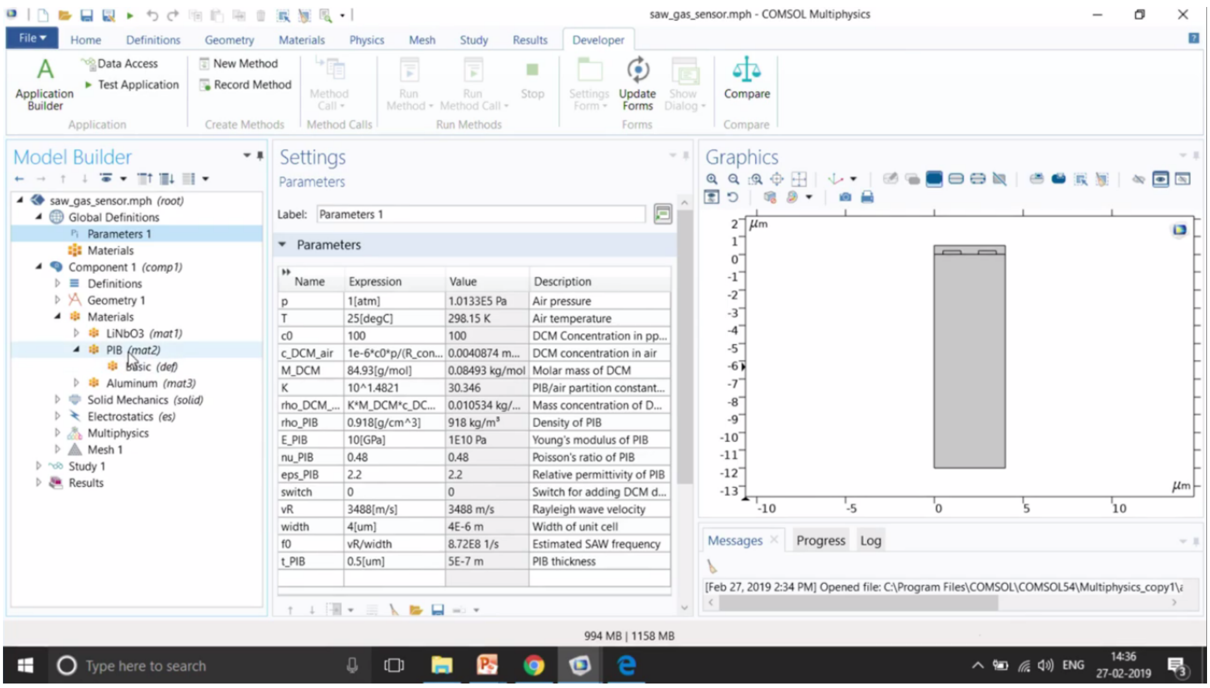
So this is the first thing that you will see once you open that particular model. So let me just open this component section. So a COMSOL has a top to down approach. So first thing is the geometry part. The next is adding the material. So one you draw the geometry you assign the particular materials. Then you add the particular type of physics that is going to happen, in this case, structural mechanics and electrostatics, and then the Multiphysics component where it couples both solid mechanics and the electrostatics and then the machine part because it’s a numerical method so we need to divide the actual domain into small parts. And then which kind of analysis you want to perform. Finally, the post processing part that is the result.

So the approach is always top to bottom over here. One more approach is in the Ribbon pane that you can see in the top where the approach is from left to right. So you first make the geometry, add the materials, add the physics, then mesh it, then study and then finally the results.

COMSOL has also introduced a new feature of developer where you can write your own code; you can write your own code and also couple it along with the actual model. So this what you can do over here.

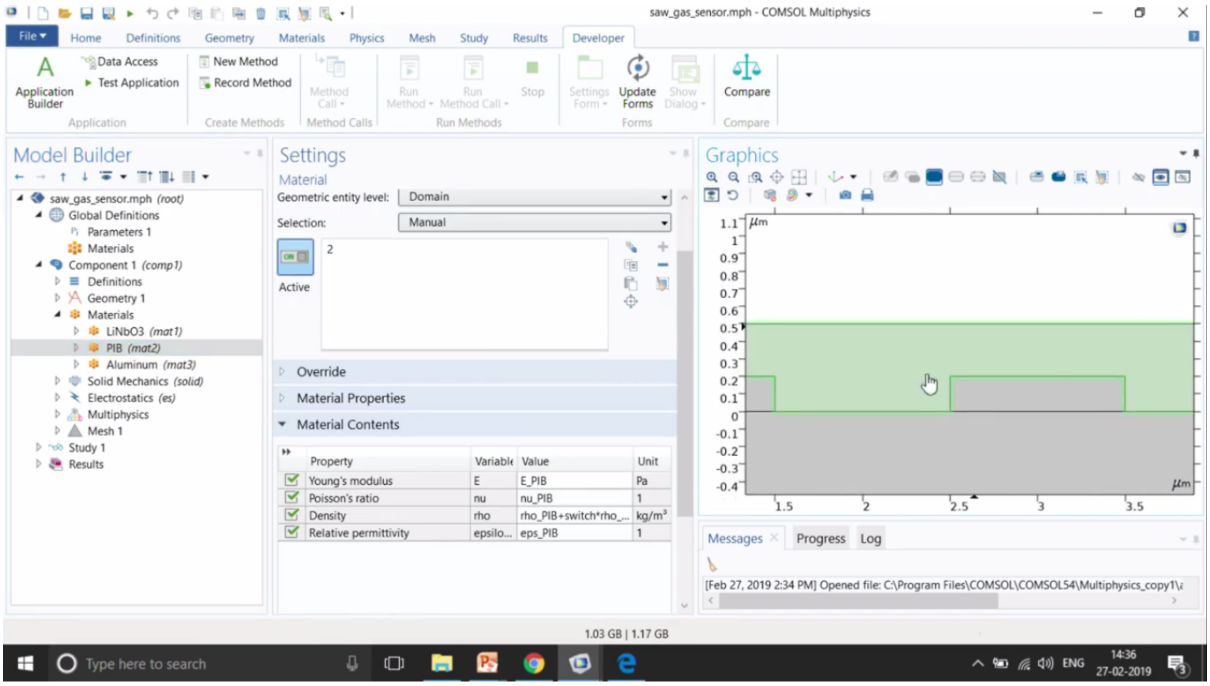
So let me go ahead and talk about the materials. So as you can see the top structure, IDT structure is modelled with aluminium over here and aluminium over here is modelled with a particular density, Young's modulus and Poisson's ratio.

Then we have the PIB domain which is modelled with a particular Young's modulus. Now you cans see E_PIB and let me just open this model and you can understand E_PIB would be defined somewhere else.

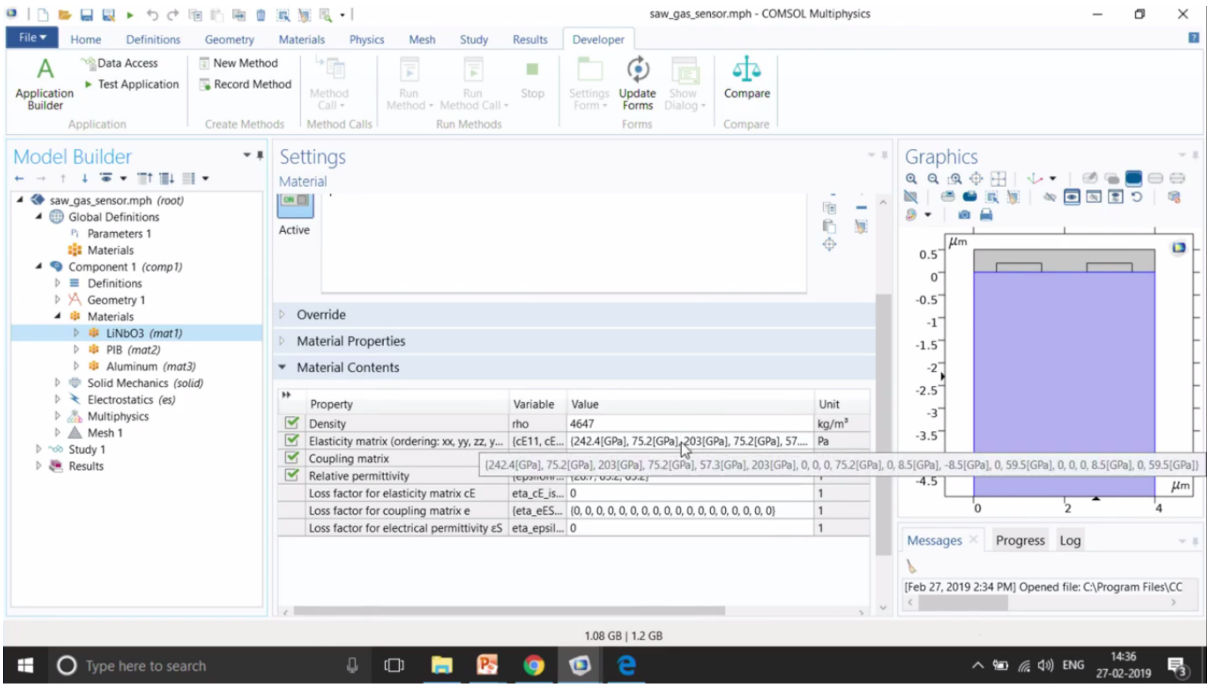


So I just go to my parameters and try to see how my E_PIB has been introduced in my modelled. So E-PIB which is a Young's modulus has been modelled as 10[GPa]. So similar to this, all the other material properties would be defined somewhere over here in the parameter section.

Classification: Public



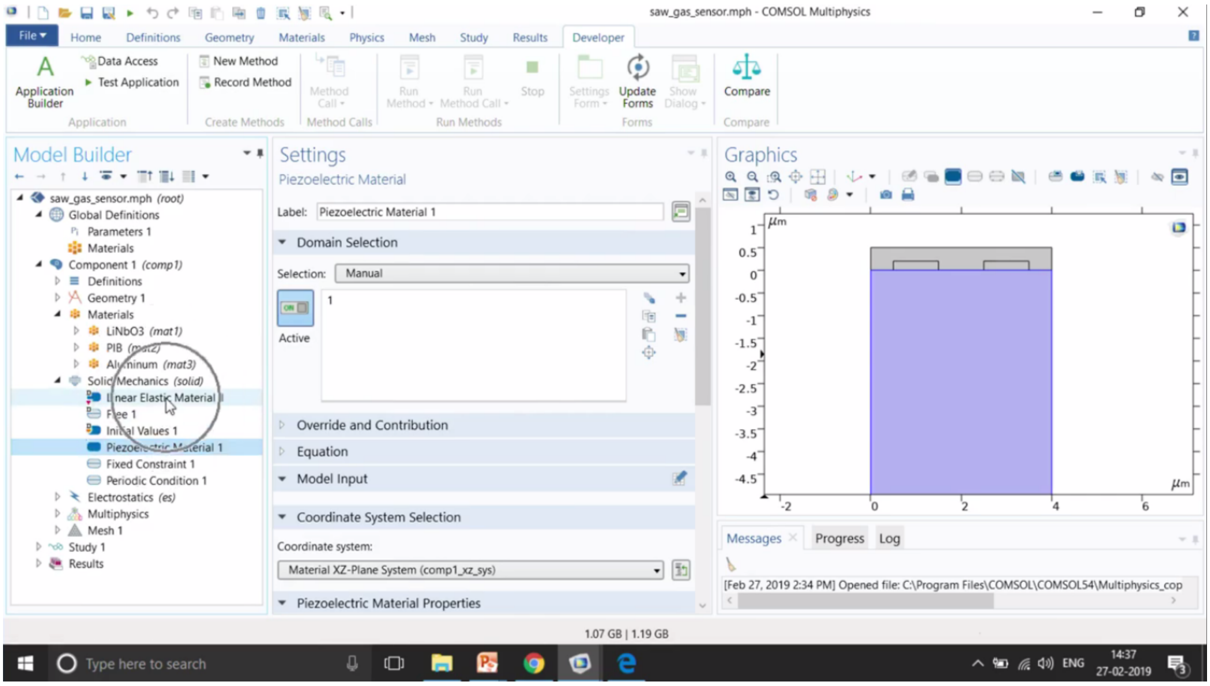
So PIB is the top thin layers that you can see over here and then finally you have the lithium niobite domain over here and this is the actual Piezoelectric domain.



That's why you can see a tenser which is defining this particular domain. So you can see a matrix of I believe it would 6x6 matrix of the elasticity matrix which couples and the coupling matrix which actually couples the structural mechanics with the electrostatic physics.

The relative permittivity as also you can see which is not isotropic material but here an anisotropic material has been used to model the relative permittivity.

So let me just go to the physics. So in this case, you can see that the Piezoelectric material, so this is something new that was introduced that you have to introduce once you make the model.



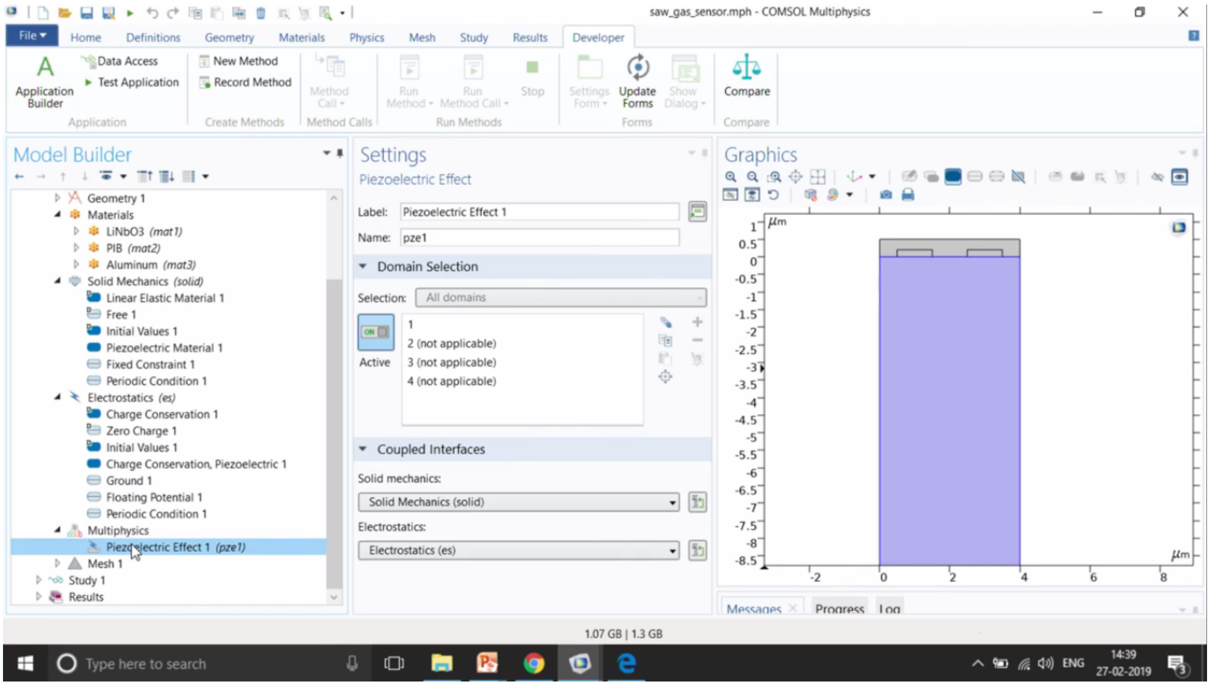
A leaner elastic material is for all the other domains but for Piezoelectric materials you need to introduce this material model. So over here, you need to write whether it is a stress charge form or strain charge form and then you give a particular fixed constraint. So here you can see that I have given a fixed constraint in the bottom.

A fixed constraint means that this is a boundary where it cannot move and then finally we give the periodic boundary condition, this is very important because we claim that we are going to do a simulation which is similar to 3D simulation but we have taken only a very small part.

How do we claim that we are doing the actual IDT structure which is a complete 3D structure. The way that we claim is using the periodic boundary condition as you can see over here. So you use a periodic boundary condition that this structure is repeating on the left and right for infinite times.

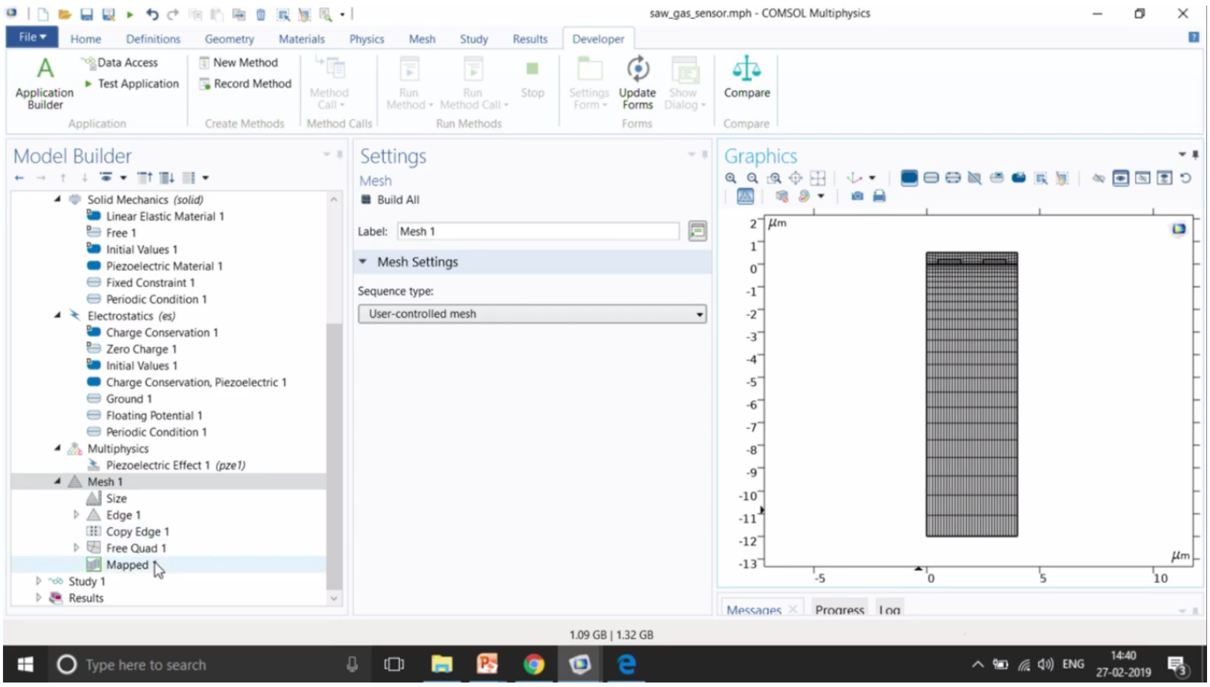
So this is again an assumption but it makes our life easy and that's why we use it. So this is how this solid mechanic structure has been modelling. With the electrostatics we need to definitely give a particular ground at somewhere. So this is the ground that we have given. So this is an IDT structure. So one would be the ground and another would be the electrode where I need to gather the voltage back.

So to gather the voltage back I am using something known as floating potential. This is very important. So if you want to give the voltage you can use the electric potential boundary condition or a terminal boundary condition to give the potential to give a particular current, but if you want to withdraw a particular voltage, you need to use a floating potential boundary condition. This is very important. And then again we are using a Piezoelectric material model for the lithium niobite oxide.

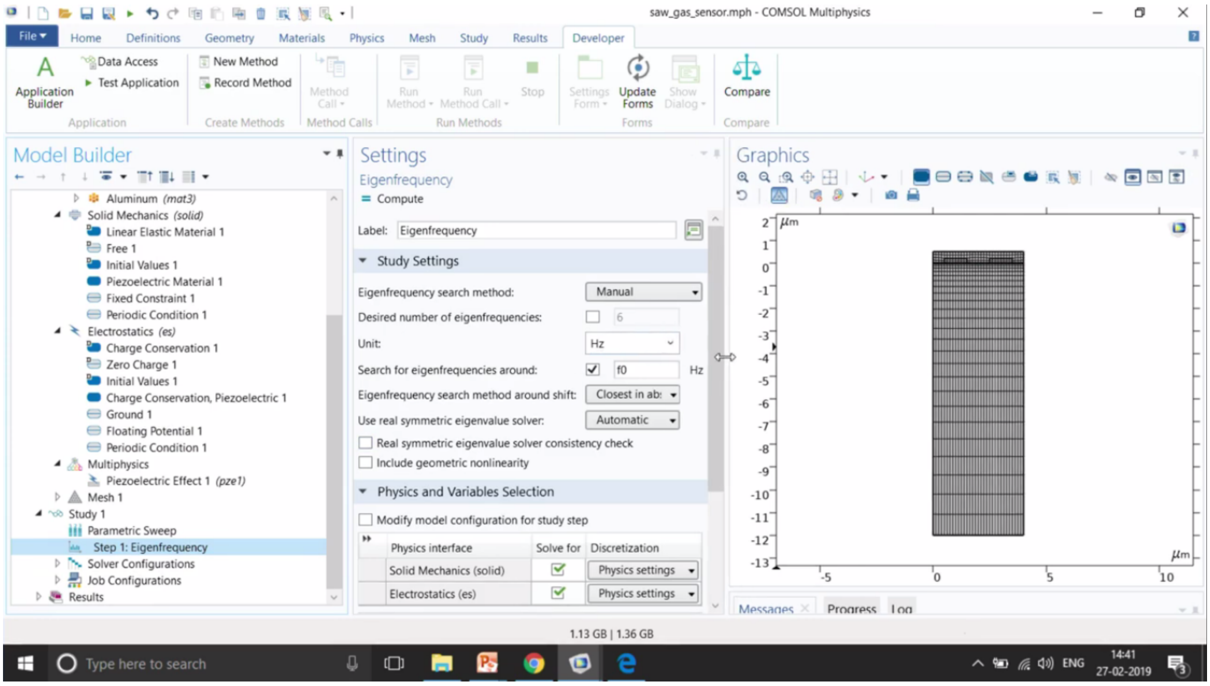


And the next part is the Piezoelectric coupling. So this is what you need to add, means this is what needs to be added that there is a coupling between solid mechanics with the electrostatics. So just to add it, just right click on it and add this Piezoelectric effect.

And then the machine part, this is also an interesting machine. So COMSOL allows you to mesh based upon your requirement. So based upon how the forces are moving you can use those particular type of mesh which is suitable for such kind of sources. For example, here the forces are more or less going to be lateral, so it is going to move from bottom to up or top to bottom. So we have used the mapped mesh.



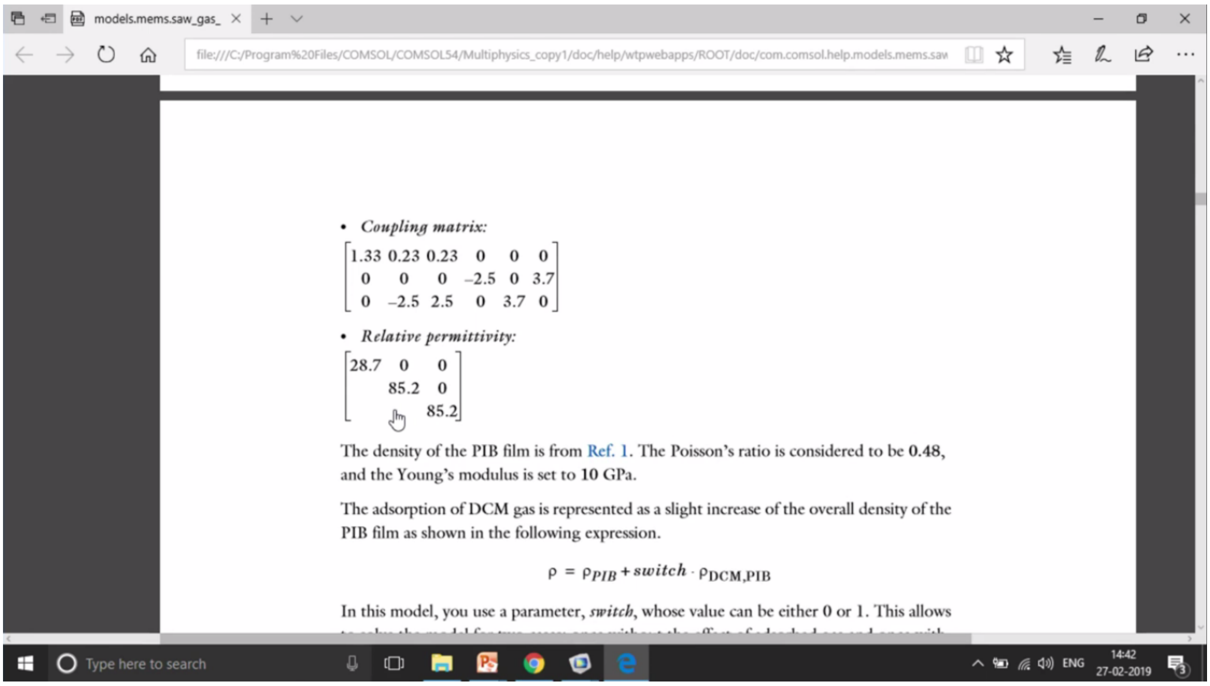
So we first tried to use the edge mesh and then copy to the right side, so this first I've meshed and then I copy it to the other side, then a free acquired mesh in the top and then I map it to the bottom. So this is a structured mesh that you are seeing right now.



The next step is the study node. So an Eigen frequency analysis figures out what are the resonant modes of your structure. So it will tell you about resonant and de-resonant modes. Over here what I've done, I know particular frequencies where the resonant frequency may lie so I have returned as f0 which I have defined in the parameters section.

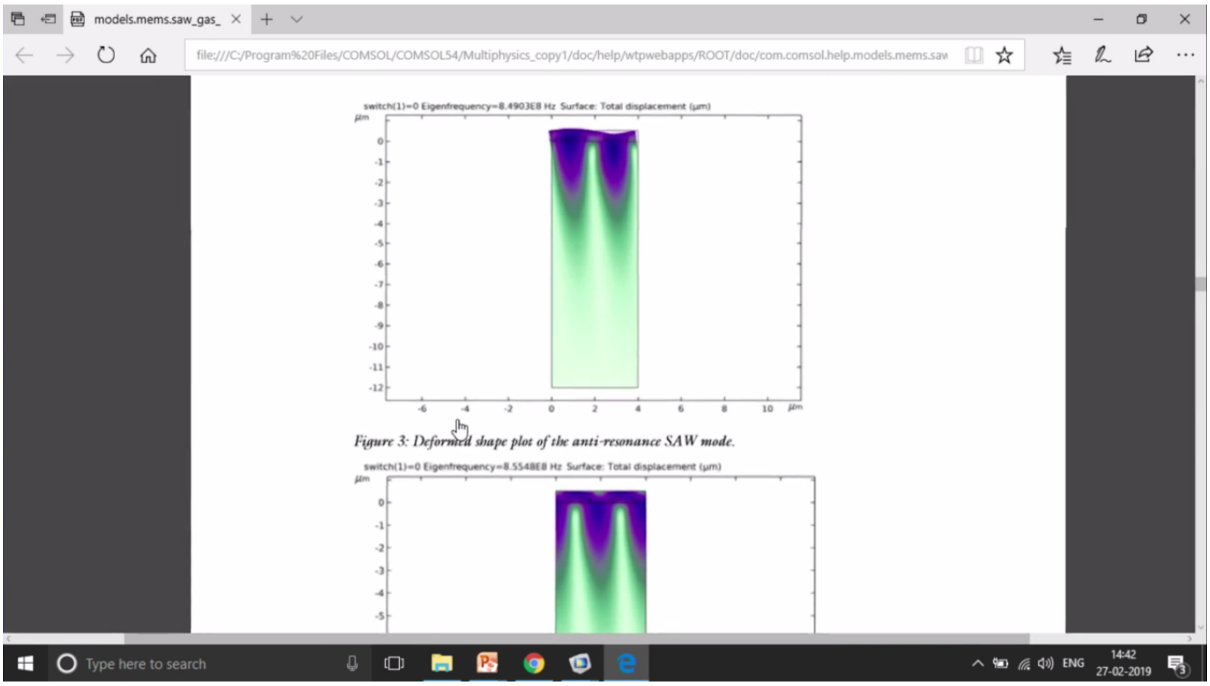
So I go to my f0 and you can see that this is a stimulated SAW frequency. So it's not that you do not need to know anything. You need to know something about your system of what frequencies would actually the acoustic waves would be existing.

So this is a basic setup and then we also do a parametric suit of switch that is 0 and 1 that is with and without the gas.



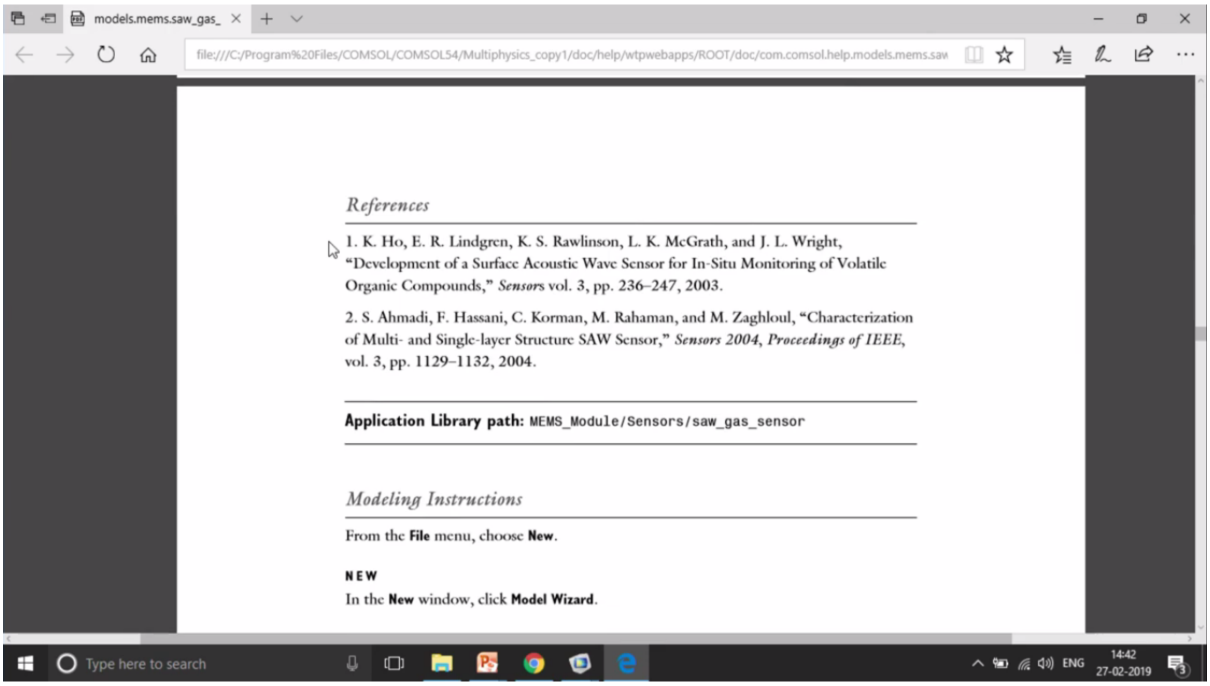
To showcase the results, so this is the actual elasticity matrix that I was talking about. This is an upper triangular elasticity matrix and this is a coupling matrix of 6x6 ad then the relative permittivity.

This is again very important because we have assumed that the relative permittivity is not isotropic. We have assumed it as a diagonal matrix, first of all, and it's also having node in x direction as compared to the y and z direction.

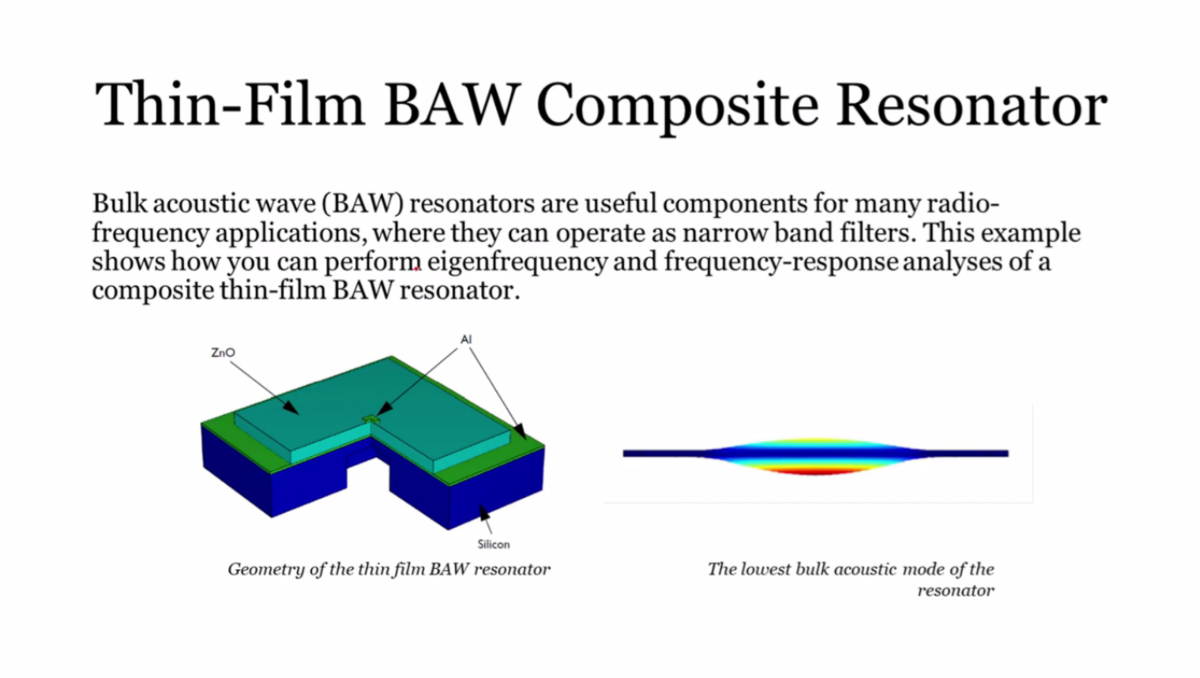


And then we go to the results. So you can see that this is an anti-resonant mode, SAW mode and this one is a resonant SAW mode. So both of them have been captured in the Eigen frequency analysis. It's the type of an analysis. You will see many other types of analysis throughout the day.

The Eigen frequency analysis figures out the resonant modes of your structure.

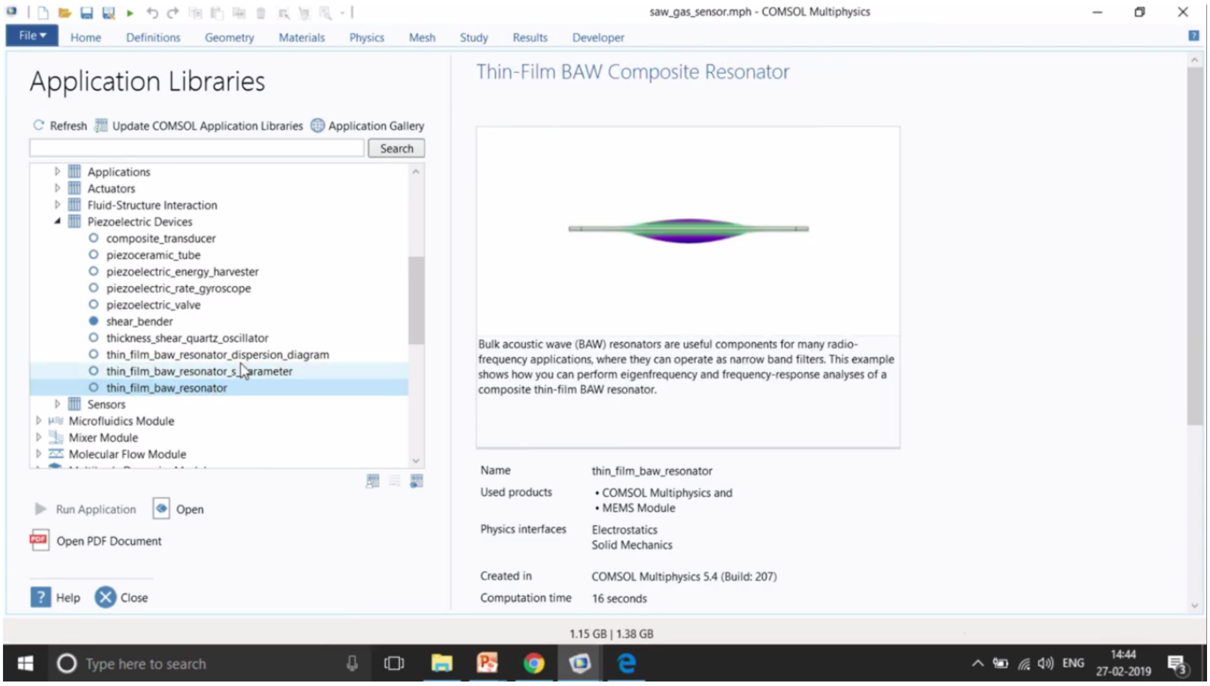


There are also references with which the results have been compared so if you want you can also go through references along with the results that you are getting in COMSOL and then you can put it with your own gas sensor design.



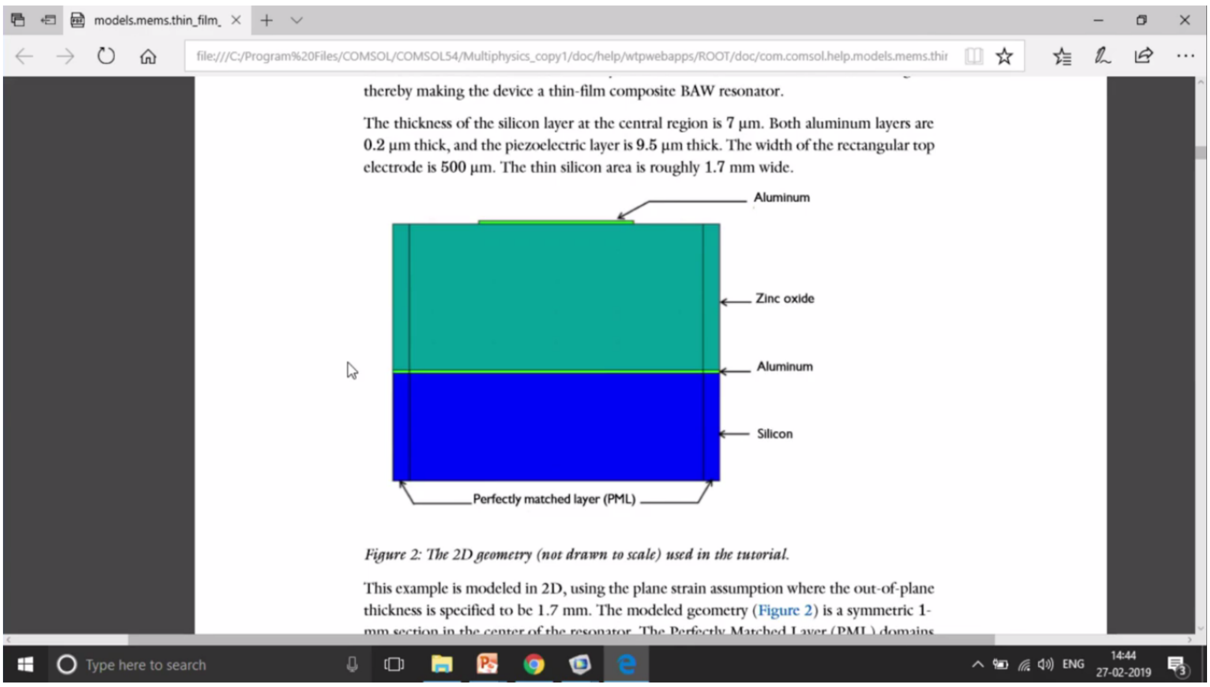
The next structure is the bulk acoustic wave composite resonator. In this particular structure again we have an aluminium domain in the top, then we have zinc oxide Peizorated material in the second layer, and then third layer is of aluminium again that is the metal contact layer and the final layer is made up of silicon that's like a huge part of, it's made up of silicon.

Again an interesting deviation from the 3D simulation would be doing a very small 2D simulation. So let's see how in COMSOL we have done this simulation.



So I go to my File> Application Library, and I again search for bulk acoustic wave. So in bulk acoustic wave, there are many different types of analysis that have been performed. We perform the dispersion diagram, we perform the AS parameters and we also perform the resonator. This is the most simple bulk acoustic wave resonator model.

So will just open the PDF document to showcase how does the modelling has been performed. This is the same diagram that we saw in the presentation. However, the way we have modelled is not the complete three domain but we have modelled only a 2Dstructure.



In addition to it as we saw in the earlier example, periodic structure were modelled. But in this case, there is no case of periodicity. It’s not about the structure of aluminium, it’s not actually getting repeated. In this case, hence we have used PMLs on the left and right. So the forces that are going to go, the electric currents that are going to flow are not going to get reflected back from these boundaries.

We should not forget that 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 mapped mesh or flip mesh.