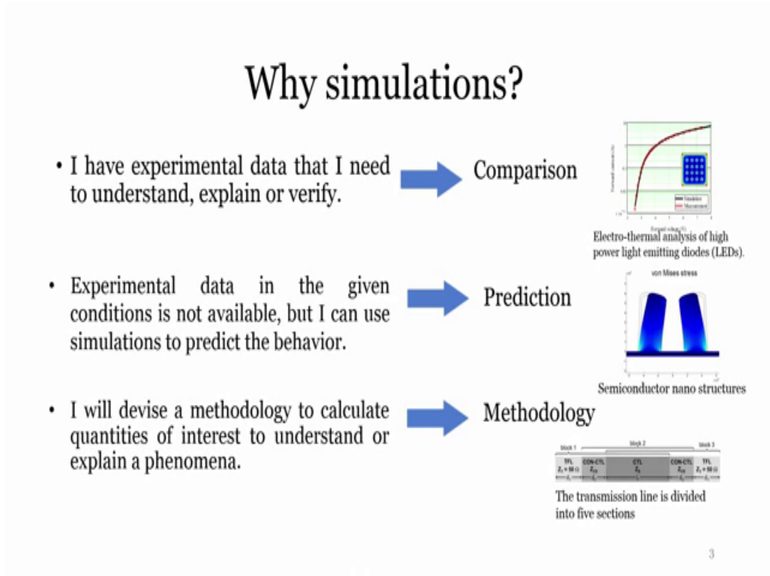
Electronic Systems for Cancer Diagnosis
Dr. Hardik J. Pandya
Department of Electronic System Engineering
Indian Institute of Science, Bangalore

Lecture – 65
MEMS Simulation Using COMSOL Multiphysics

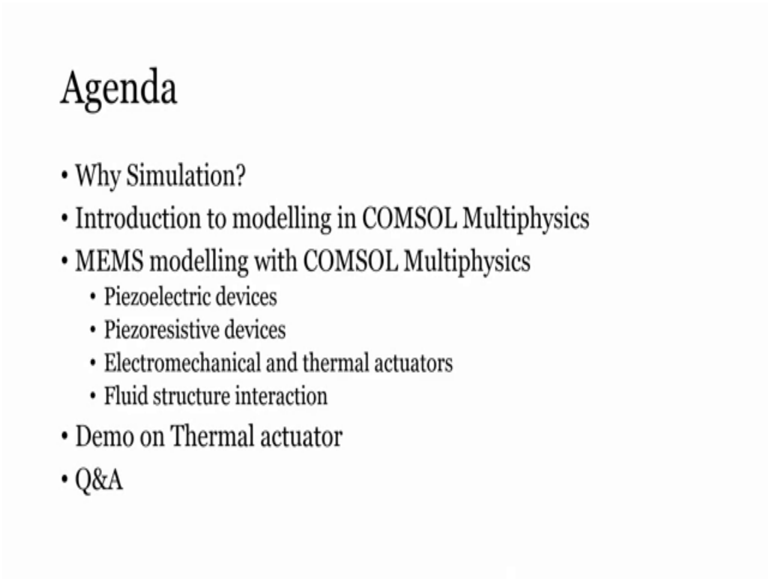
Hi, welcome to this particular lecture this lecture is focused on MEMS Simulation Using COMSOL Multiphysics. Now, you see why you need to learn a simulation right for this particular course. We have seen a fabrication of sensors right, but there are a lot of questions, then we need to address before we fabricate the sensors. And simulation will help us to answer those questions.

(Refer Slide Time: 01:19)



So, some of the questions can be if you have the experimental data right and need to understand or verify those data, then you can use this particular simulation software. Second can be if experiment data are given and the conditions are not available, I can use the simulation to predict the behavior. While the third can be if I devise the methodological quantities of interest to understand or explain a phenomenon, then you can use the simulations.

(Refer Slide Time: 01:41)



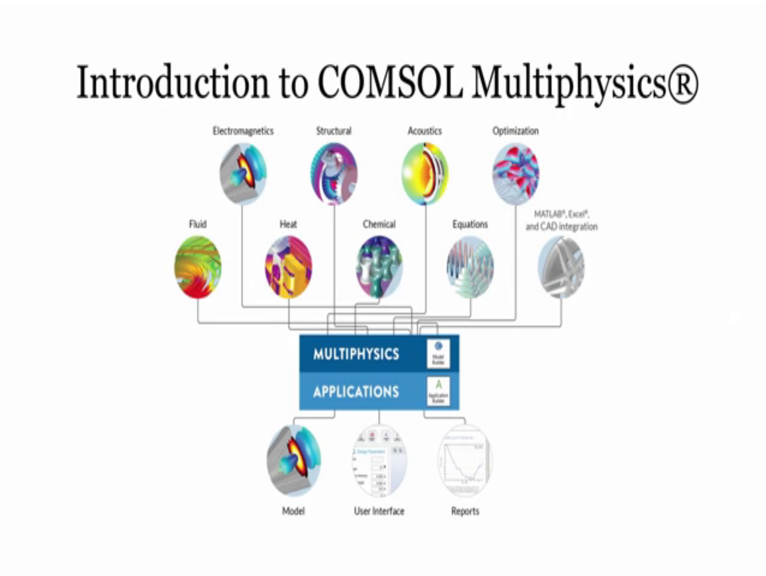
So, if I see why exactly simulation is required, then these are the three answers to the question that we can design or simulate the device if the given data is, if you have given data set of data. Second one is to understand how device will perform if you have fabricate using the conditions that you had determined right. And that and the third one is to understand the methodology is correct or not right. So, these three things are there.

Before we go to this particular thing let us see what will be you know talking in this particular lecture, we will be understanding why we require simulation which we have just seen three different reasons. Introduction to modeling in COMSOL Multiphysics and then we will see few of the examples of MEMS based modeling with COMSOL Multiphysics. We will talk about piezoelectric devices, we will talk about piezo resistor devices, we will talk about electromechanical and thermal actuators, we will also talk about fluid structure interactions. And then we will actually see the demo on thermal actuator following by some question answers if you have, you can always ask us on the forum.

So, let us move to this particular slide. Now, for example, if I have the experimental data, then how can I explain or verify. So, if you have to compare, so for example, you can do the measurement when simulation. For example, ETA Electro Thermo Thermal Analysis of high power light emitting diodes, you can you do a comparison with whatever data you have obtained with these simulations.

Second one is when you understand the nanostructures, the stress in the nano structure if you want to predict that how much would be the stress on applying a certain amount of force, then you can understand or use the simulation. Finally, if the transmission line is divided into five sections, you can understand whether your methodology or quantity of interest to understand particular phenomenon is correct or not, your methodology to design the device is correct or not. So, for that you require simulations.

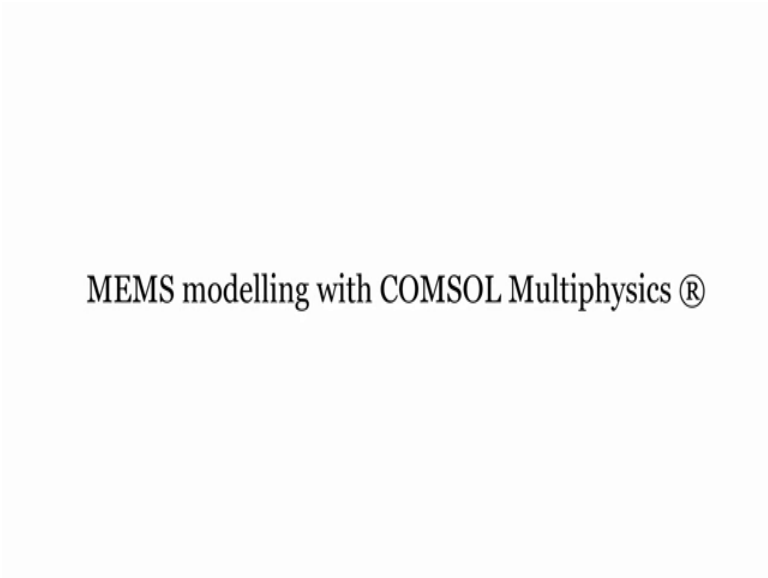
(Refer Slide Time: 03:53)



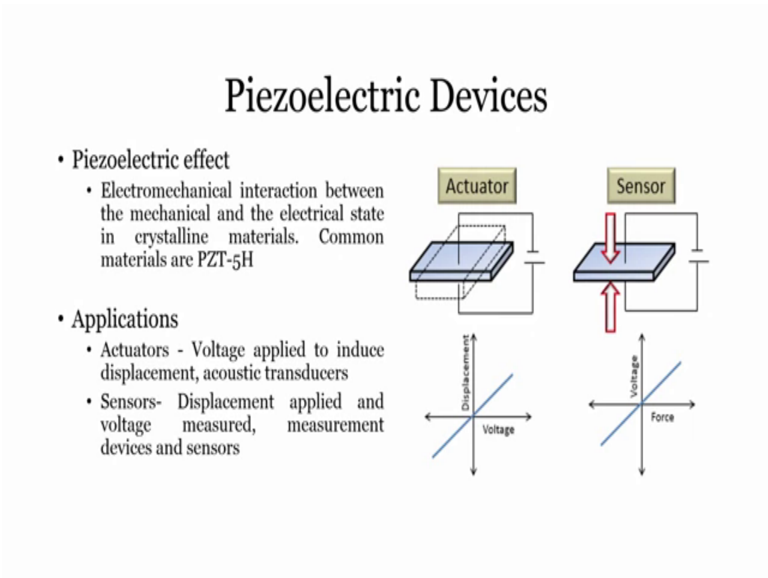
So, when you talk about COMSOL Multiphysics, we will be discussing the details in one of the class about how to use this software. But for you to understand how what are the applications and why where you can use this particular software. The first one is in fluid dynamics. You can use to understand the macrofluid system and the fluid flowing through the particular channel, you can use in electromagnetic, you can use in heat. We just know if you take an example of a heater you can easily understand.

In structural thing, for example, if there is a cantilever and you want to know the stress and strain that cantilever. In terms of chemicals what chemicals you can use is the in that particular software. In terms of acoustic sensor, if you want to understand whether the acoustics sensor that your design suppose a diaphragm whether it will work correctly or not, to verify your equations for optimization as well as for integrating the MATLAB and Excel and CAD right. So, this is the advantage of COMSOL multi physics. Of course, you can do multi physics with applications for example, you can have the model and then you can use the interface to finally generate the reports.

(Refer Slide Time: 05:05)



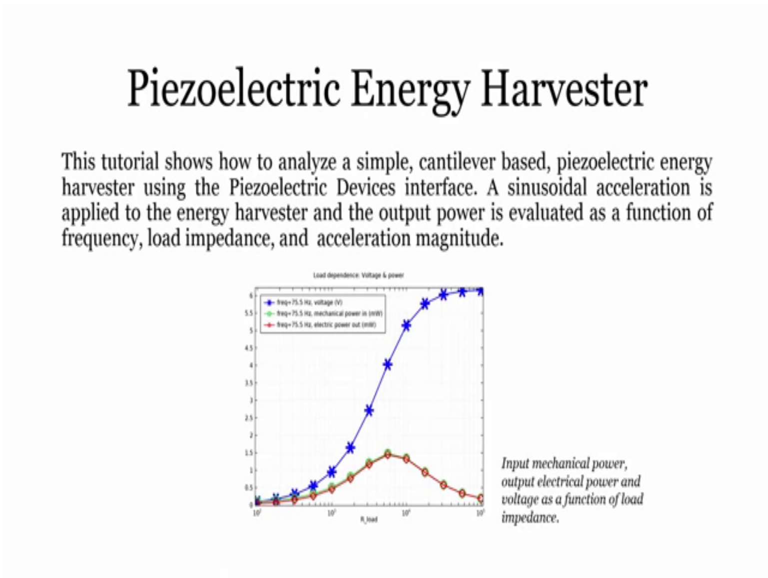
(Refer Slide Time: 05:09)



So, like I said we will be discussing of piezoelectric devices. Let me just go quickly through the piezoelectric devices, and then in the experimental class we will see in detail how these electric piezoelectric devices can be used with the given COMSOL Multiphysics software. Now, when you talk about piezoelectric effect that this is an electro mechanical interaction between the mechanical state and the electrical state in the crystalline materials, you know that when I apply pressure on a piezoelectric material, there is a change in the voltage right.

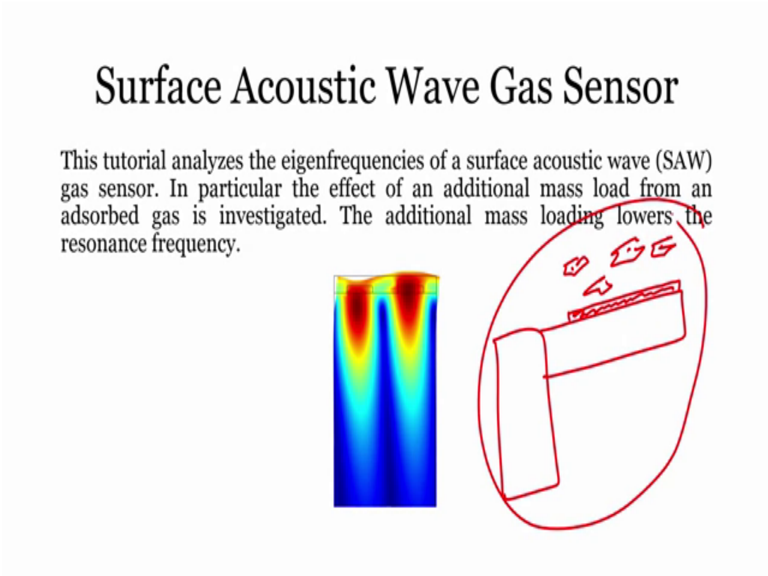
There can be many applications to piezoelectric material, for example, it can be used as actuator for understanding the displacement, in terms of acoustics transducers, it can be used for the displacement applied for a many sensor measurement of devices and sensing a lot of applications are there for piezoelectric material.

(Refer Slide Time: 05:59)



It can also be used as the energy harvested. And we will be looking at the tutorial on how to analyze a simple cantilever based piezoelectric energy harvested.

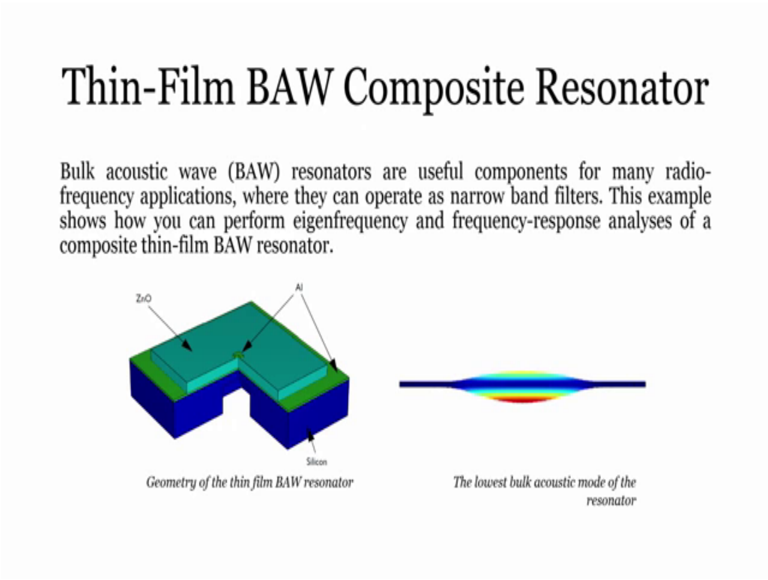
(Refer Slide Time: 06:11)



Then we will also see surface acoustically gas sensor. So, in which the tutorial will be analyzing the Eigen frequencies of our Surface Acoustic Wave that is SAW gas sensor. And in particular we will see the effect of additional mass load from the absorbed gas. So, for a given material even when the gas is absorbed, there will be changed in the mass and this mass lowers the resonant frequency. So, suppose I design a cantilever I said this is a silicon wafer and I have cantilever all right, then this cantilever will vibrate at particular frequency.

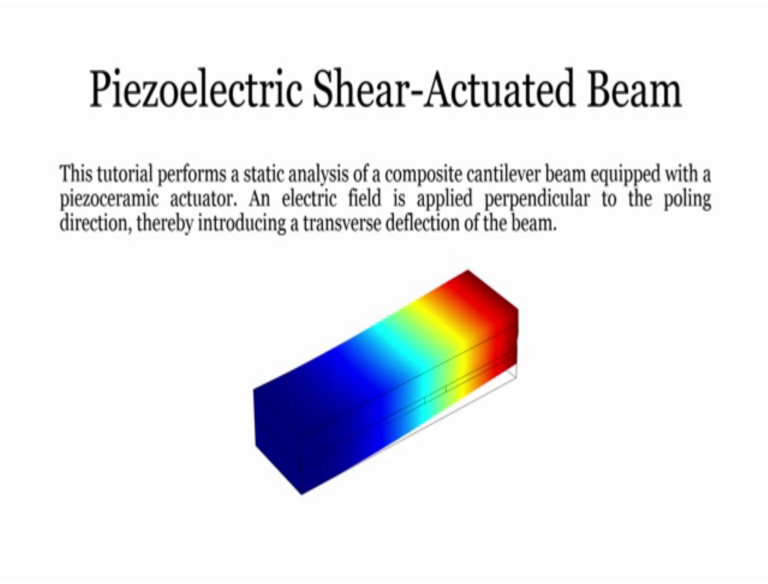
And if I was sensing material on the cantilever, when there are gas molecules like loaded in this cantilever or onto the cantilever absorbed in the cantilever, the frequency the resonance frequency would be different. That resonance frequency we can understand the or we can simulate it with the help of the COMSOL Multiphysics.

(Refer Slide Time: 07:10)



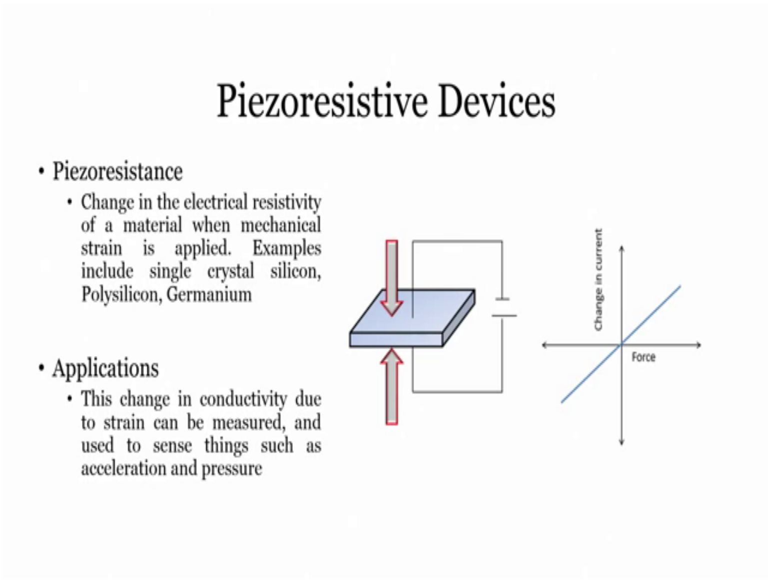
We can also design and simulate bulk acoustic wave resonators which are useful components for many RF applications all right. In particularly, when they can operate as a narrow band filters. We will discuss this thing in detail.

(Refer Slide Time: 07:25)



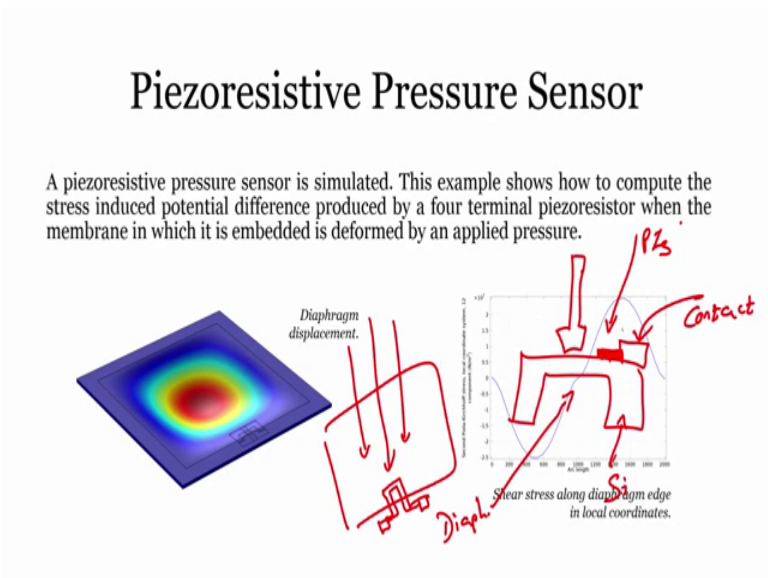
We will also understand the piezoelectric shear actuated beam. And, in this tutorial particularly a static analysis of composite cantilever beam equipped with a piezoceramic actuator will be taught.

(Refer Slide Time: 07:39)



Then we will see piezoresistive devices. Now the difference between piezoelectric and piezzoresistive is that. In piezzoresistive devices when we apply a pressure, there is change in resistance. Well in case of piezoelectric on the same thing when we apply a pressure or force, then that is change in the voltage electrical signal. So, the example of a piezoelectric devices are silicon, poly silicon, germanium. While the applications are that the change in the conductivity due to strain can be measured to use the use to sense things such as acceleration and pressure. There are a lot of other applications in terms of biomedical domain, lot of application in terms of electronics domain.

(Refer Slide Time: 08:23)

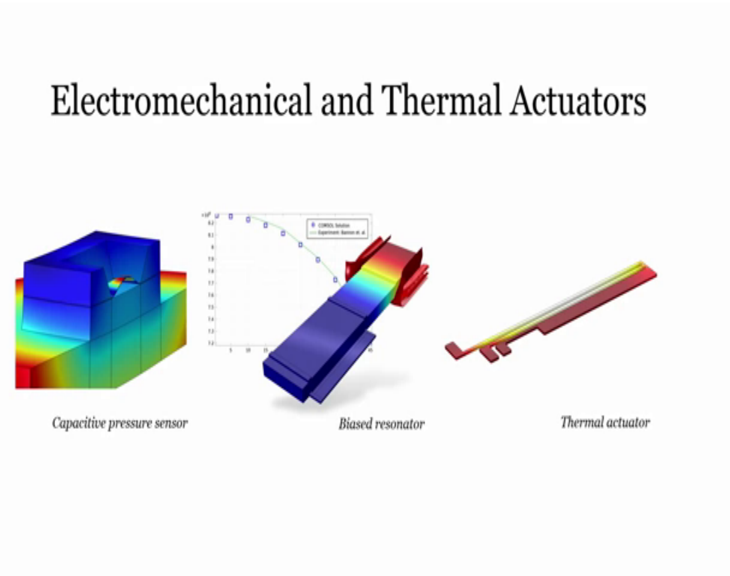


If you know the use of COMSOL, it can make your life easier ok. Now, if I have a diaphragm right and I want to understand if I apply a force what is a change in diaphragm? For example, piezoresistive pressure sensor. If I have let us say this is the silicon wafer and I designed my sensor that is sensor is here on this one. And I am applying a pressure or force on this particular chip.

Now, we understand if I draw cross section, the cross section is like this and the sensor is here and the edge. If I at this is my sensor ok and this is a contact to the sensor right this is a contact. This is my piezoresistive sensor. This is silicon wafer; this is a diaphragm. If I apply a force on this particular diaphragm, then this diaphragm displacement and that diaphragm displacement causing the change in the piezoresistor can be simulated with the help of COMSOL Multiphysics you see.

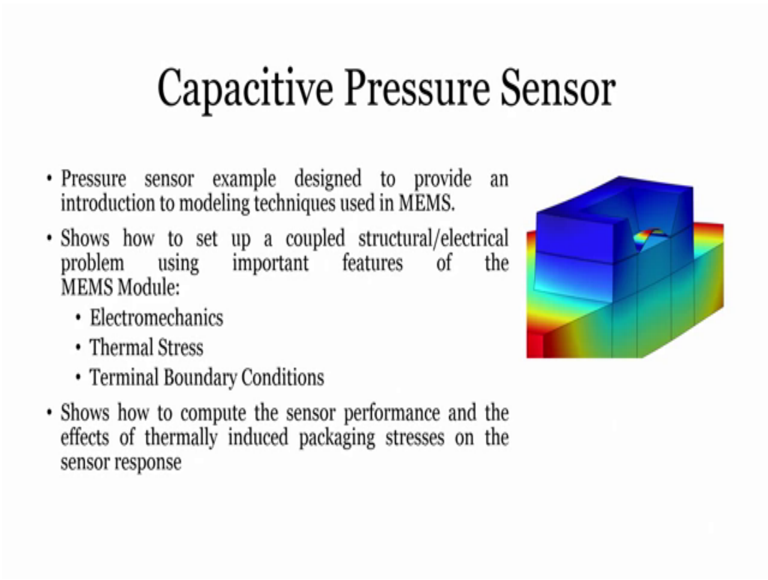
So, this is nothing, but a pressure sensor. If I apply a pressure and if I see the change in resistance is nothing but I can design my pressure sensor with the help of piezoresistive material right, so that is the advantage of using the simulation software.

(Refer Slide Time: 10:09)



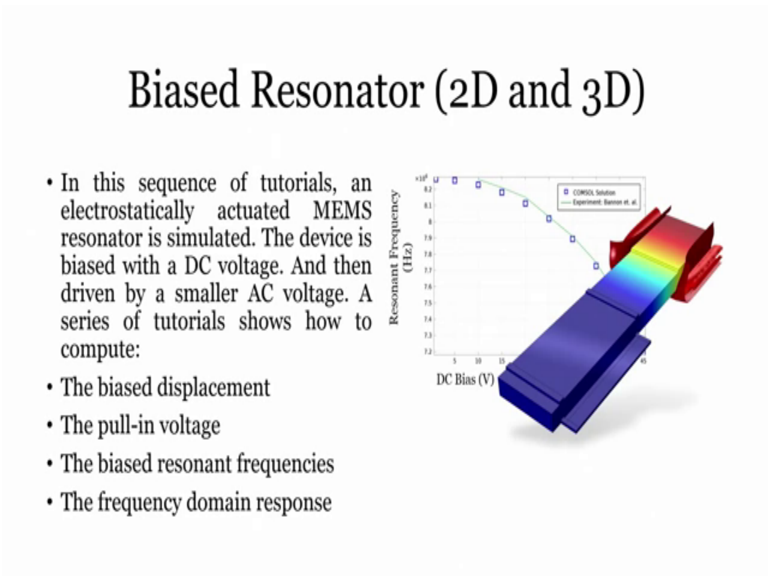
We will again see in detail in the TA class where we will also understand the capacity pressure sensor, the biased resonator as well as the thermal actuator in detail in the following lecture.

(Refer Slide Time: 10:21)



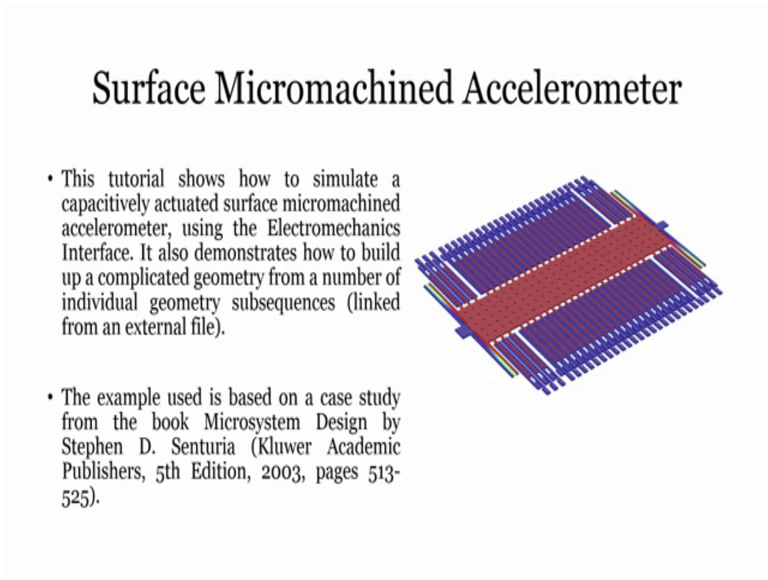
How the capacity pressure sensor would be used and how to setup a compiled structural electrical problem using important features of MEMS module such as electro mechanics, thermal stresses, terminal boundary conditions, this everything we can design with the help of this particular simulation software.

(Refer Slide Time: 10:43)



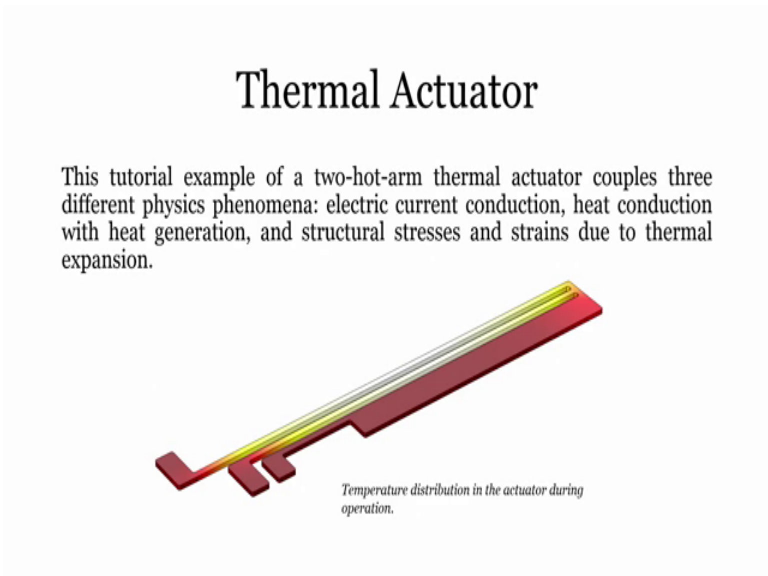
We will also see how the biased resonator can be used, and how the pull in voltage by the frequency, frequency domain response occurs. And we will also understand the micro machine accelerometer.

(Refer Slide Time: 10:51)



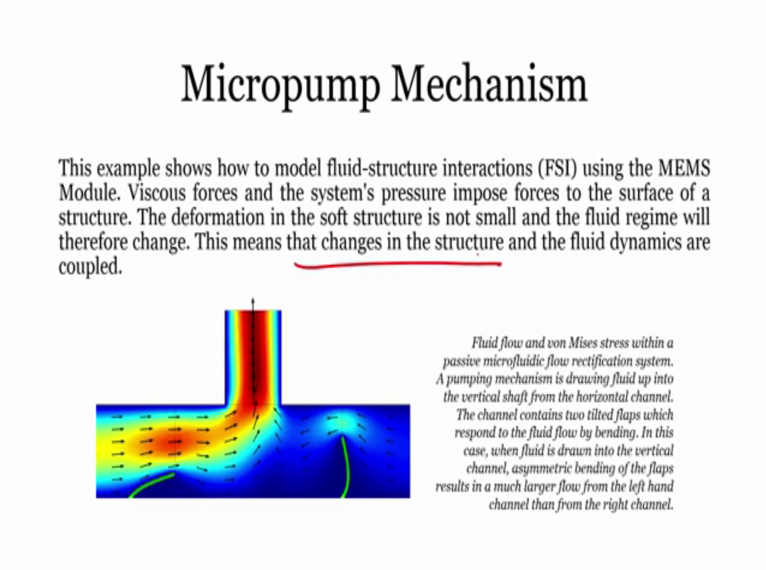
How we can design a or simulate a micro machine accelerate accelerometer. And the example for this particular thing is based on the case study from the book Micro System Designed by Stephen.

(Refer Slide Time: 11:07)



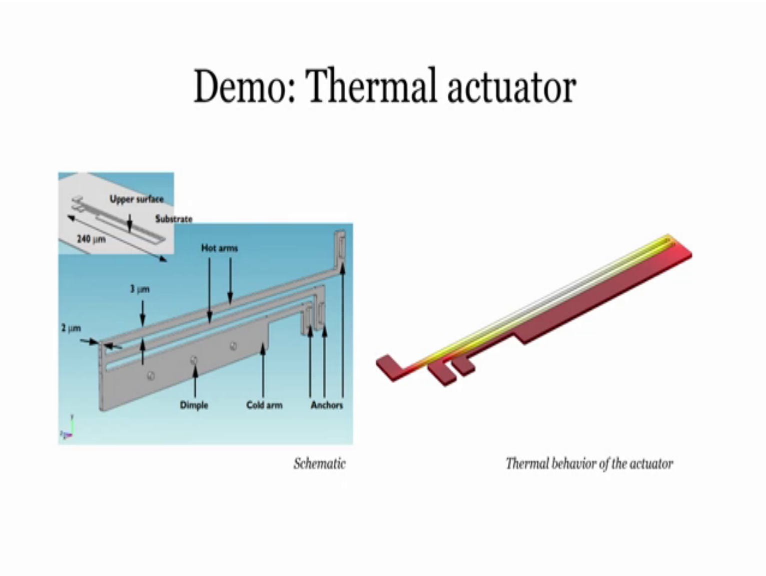
While we will also design the thermal actuator in which we will take an example of two arm two hot arm thermal actuator and that will coupled with a three different physics phenomenon such as electric current, conduction, heat conduction band width, heat generation ok, so along with structures stresses and other things. Same way if you want to understand the design of the microfluids, you can also use the COMSOL Multiphysics in detail, where you understand a fluid structure interaction do it from the when the fluid is flowing in the particular in a micro fluidic channel right.

(Refer Slide Time: 11:45)



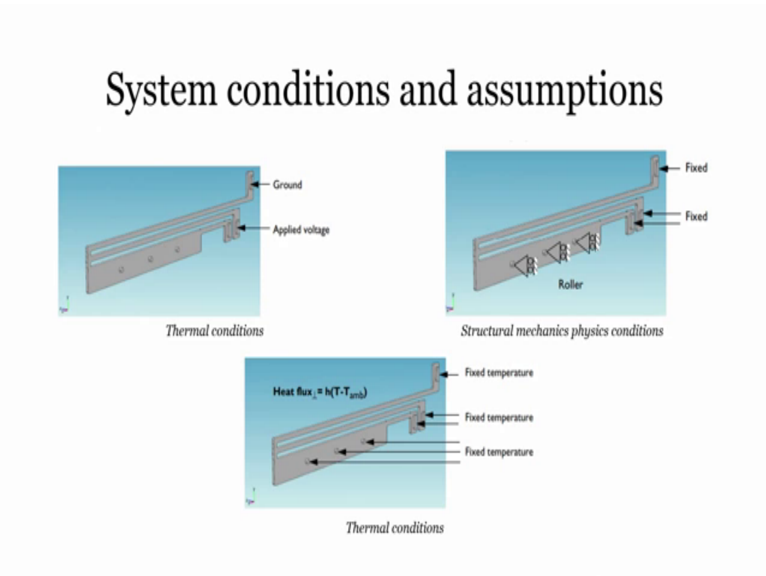
And same thing we will understand how exactly the micropump mechanism would be used. We will say that this particular example we will see how to model the Fluids Structure Interaction which is FSI using this module. We can understand viscous forces and the system pressure imposes forces to surface of a structure also we said that deformation in the soft structure which is not really small is not small the fluid and will there for change this means is change in the structure and fluid dynamics are coupled with this. Why we can say that these changes in structure and the fluid dynamics are coupled, we will discuss in detail.

(Refer Slide Time: 12:21)



We will send a thermo of the demo of a thermal actuator and how we can you know design this particular actuator.

(Refer Slide Time: 12:27)



And then we will end up the TA class on that particular lecture note. So, the idea of showing you the COMSOL Multiphysics and to take a lot of application is to help you that before you go for fabrication. Suppose you take the help of INUP in sense that we have in ISE. And, if you go to the to that particular program, if you want to fabricate a device, before you do the fabrication if you have finished the simulation it will be easier for you to design the process flow accordingly right.

Thus understanding simulation using Multiphysics COMSOL Multiphysics can help you out and to reduce the, you know cost in a way because you know the performance or you have simulated the device performance. And, when you fabricate a device you can compare that performance with the fabricated one right.

So, having said that let us see in the experimental class how this simulation can be used. Till then I will end up my this particular introduction to COMSOL Multiphysics here and learn this as an example so that you can design a lot of sensors and transducers before you fabricate the device right. Again we have a fab lab right over here, but it is always advisable to use a simulation to get the results and then to move for the fabrication all right. So, if you have any questions, feel free to ask, till then you take care.

And I will see you sometime later all right. Bye.