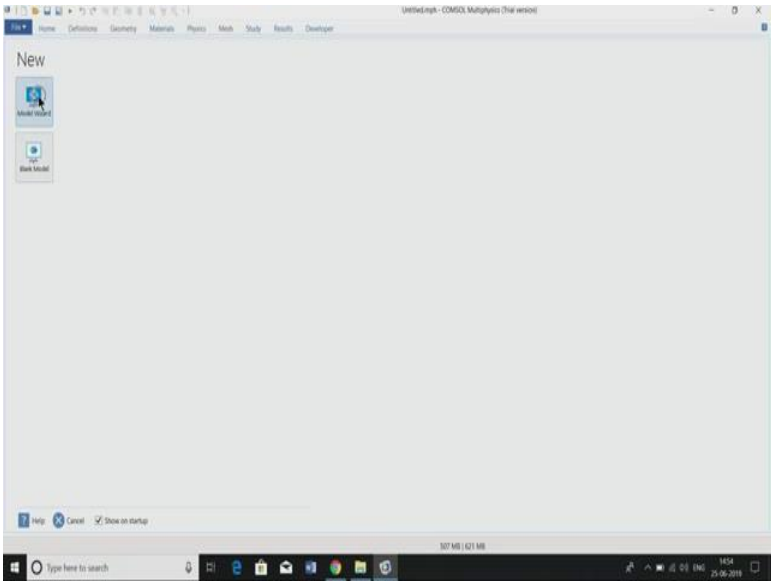
Sensors and Actuators
Dr. Hardik J. Pandya
Department of Electronic Systems Engineering
Indian Institute of Science, Bengaluru

Lecture – 31
Demonstration of Microheaters in COMSOL Multiphysics

Hi, in this class we will be showing how can you use the COMSOL Multiphysics to perform simulations with a micro heater, right. So, here my teaching assistant Anil will show it to you the simulation how to use simulation tool. Again, COMSOL multi physics is extremely important simulation tool there are other tools as well, we prefer to use COMSOL multiphysics and that is why we are teaching you this particular software and we will be looking at how can you simulate the heater. If any questions feel free to ask through the NPTEL forum till then you take care, I will request Anil to take the class bye.

(Refer Slide Time: 01:09)

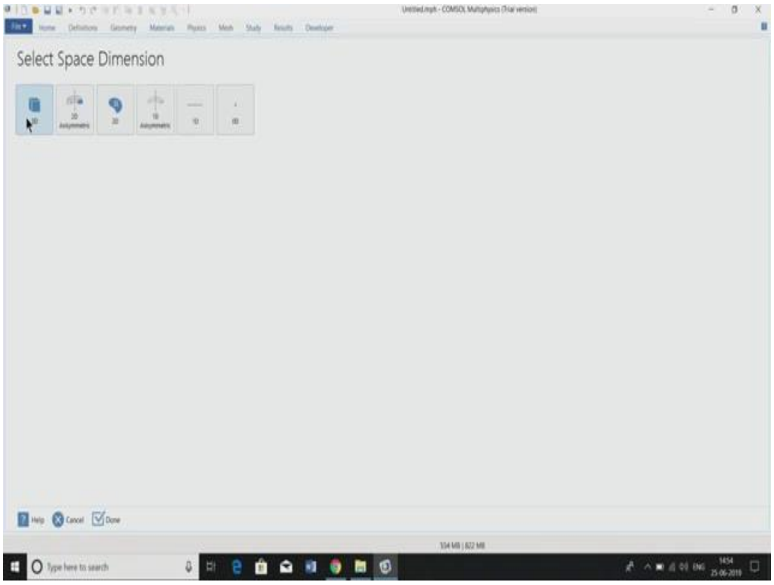


Hi everyone, welcome to this lecture, in a previous lecture you have seen an introduction to COMSOL multiphysics and how the software can be used to simulate lot of MEMS devices and other sensors, and sensors and actuators. In this module we will be seeing very small basic COMSOL simulation where you will understand the importance of doing simulations to design your devices as well as how to go about doing multi physics simulations, most of the MEMS devices. Because they talk to the real world and the sense physical parameters and they are also sometimes used to actuate other physical parameters. So, they also sense physical parameters from the real world and also actuate the physical world.

So, how can we do? So, most of the most of the time it will involved multiple physics like thermal related stuff will be there, electrical related stuff will be there, mechanical related stuff will be there. So, how can we do a wholesome simulation or a coupled simulation of multiple physical processes? So, we will see a very basic example of how can we do that and now. So, we are using trial version of COMSOL, here COMSOL 5.4 and let us try to see. So, what we going to do today is about we will be simulating micro resistor beam, what is it? I will do I will show it you why we do the simulations.

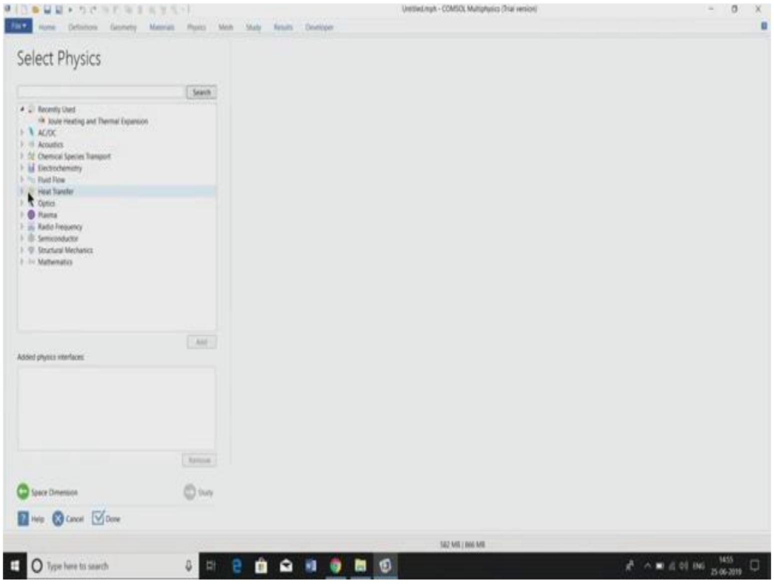
So, as and when we open up the COMSOL software first thing that comes up is this screen. So, when you open the software what we see is this window, you will have the blank model and modal wizard.

(Refer Slide Time: 02:56)



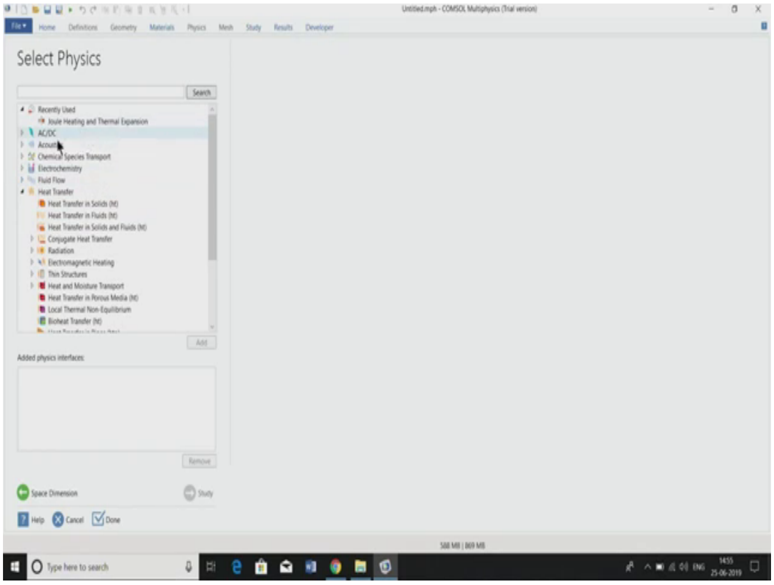
So, for what we are going to do today, you have to go to modal wizard go to modal wizard and we will be doing 3D model actually. So, in any MEMS device that you do, it will be a 3D model may be one dimension might be very less that you can approximated to a 2Ds dimension, but fundamentally it will be a 3D model. So, we have to select the 3D model. So, select this 3D model.

(Refer Slide Time: 03:18)



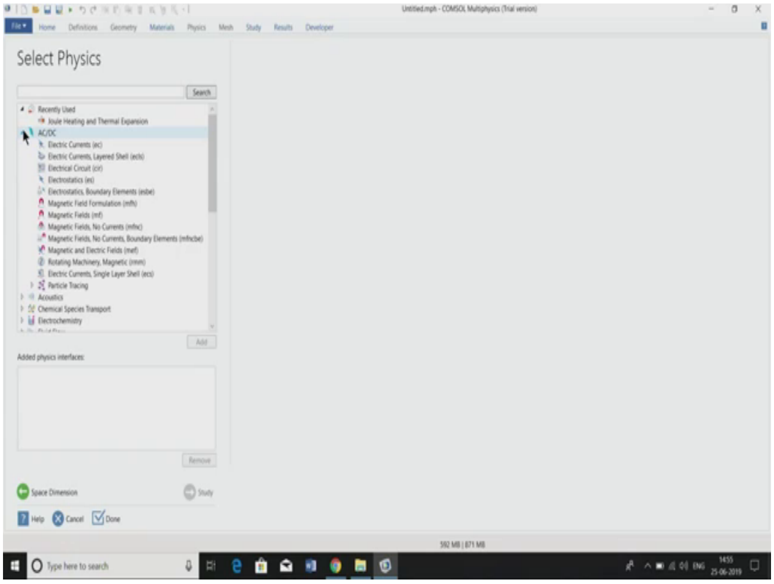
Once you select 3D model, this is the most important window. This is where we have to select the type of physics that you want to study through our design; this is where we select the type of physics. So, what are we going to do? We are going to see a micro resistor beam.

(Refer Slide Time: 03:40)



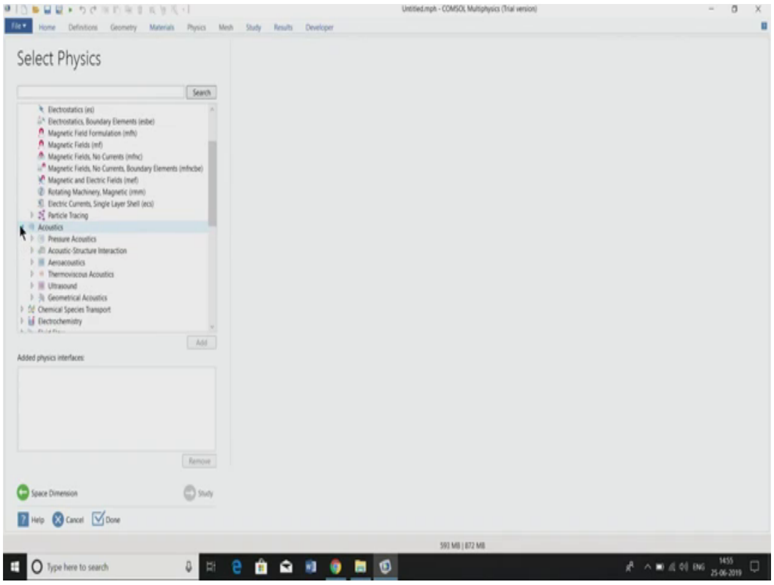
So, for that what we have to do is, go to heat transfer, we will see what are physics are there here.

(Refer Slide Time: 03:52)



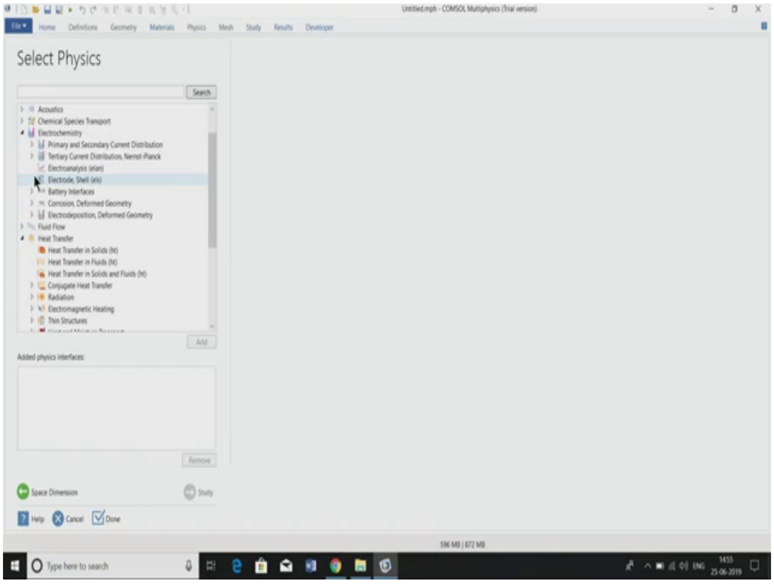
So, there is AC DC for electric currents you can see this.

(Refer Slide Time: 03:56)



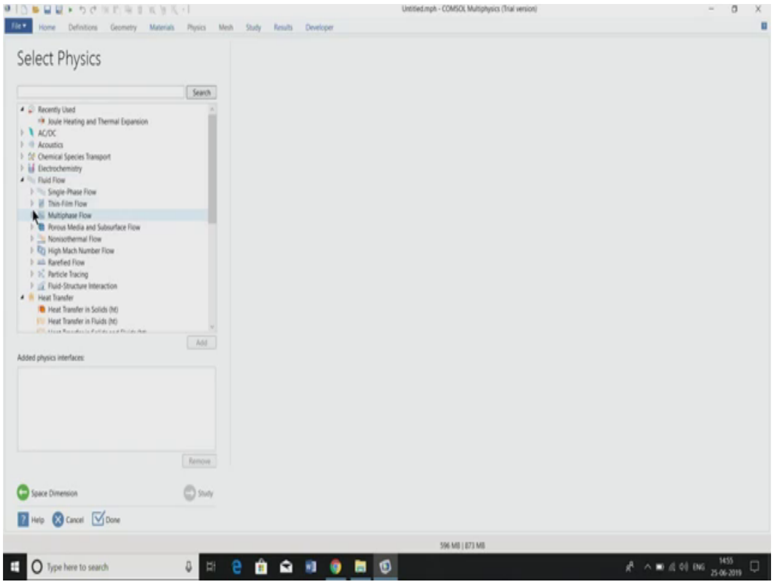
There is acoustic simulation let me collapse it back.

(Refer Slide Time: 04:01)



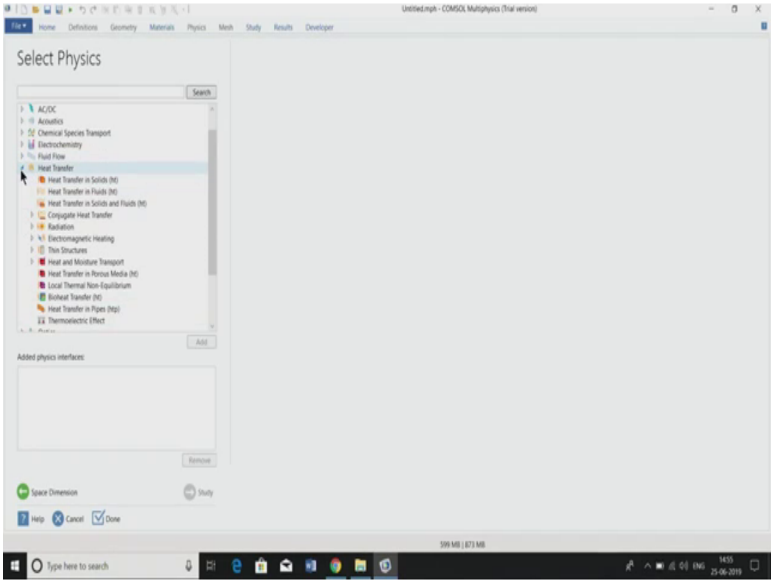
There is electrochemistry.

(Refer Slide Time: 04:10)



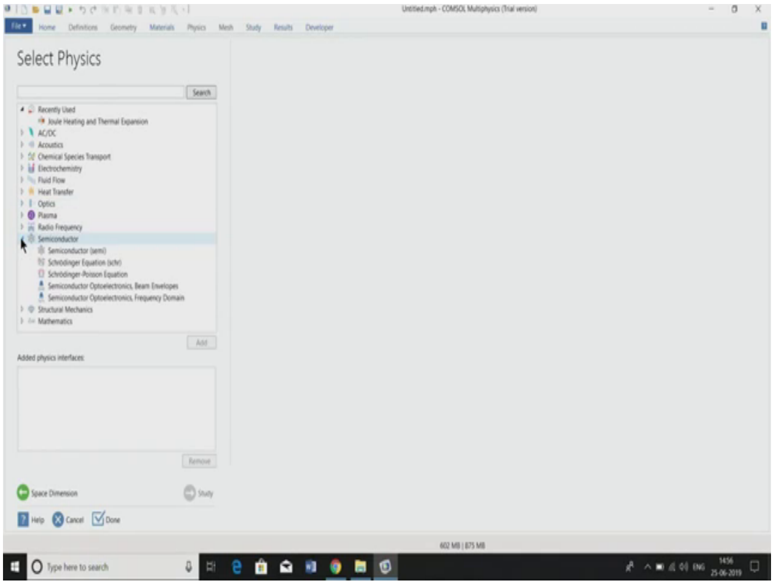
There is fluid flow.

(Refer Slide Time: 04:16)



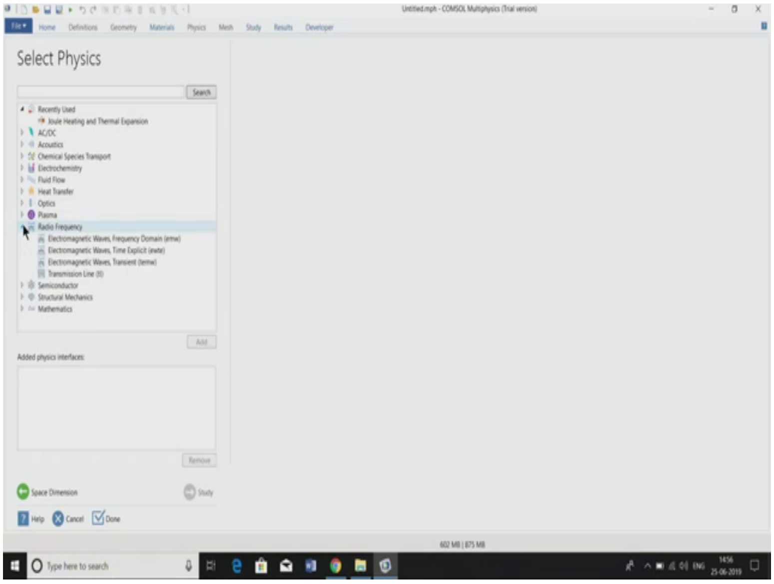
There is heat transfer. So, these are all individual physics.

(Refer Slide Time: 04:22)



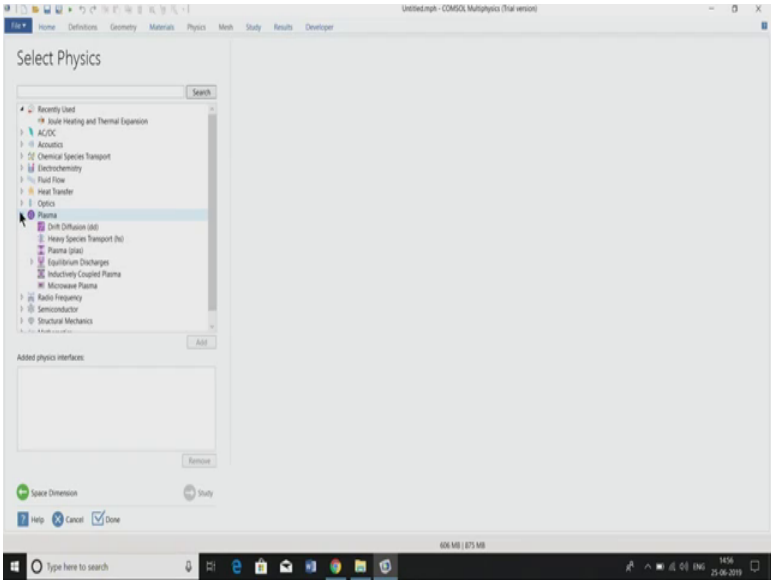
Now, what we have to do is, let us there is semiconductor simulations separately.

(Refer Slide Time: 04:24)



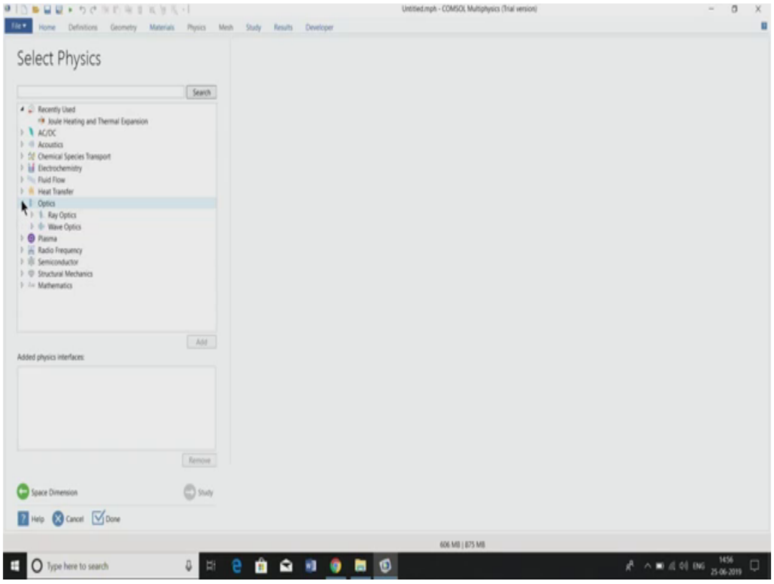
Radio frequency simulations.

(Refer Slide Time: 04:26)



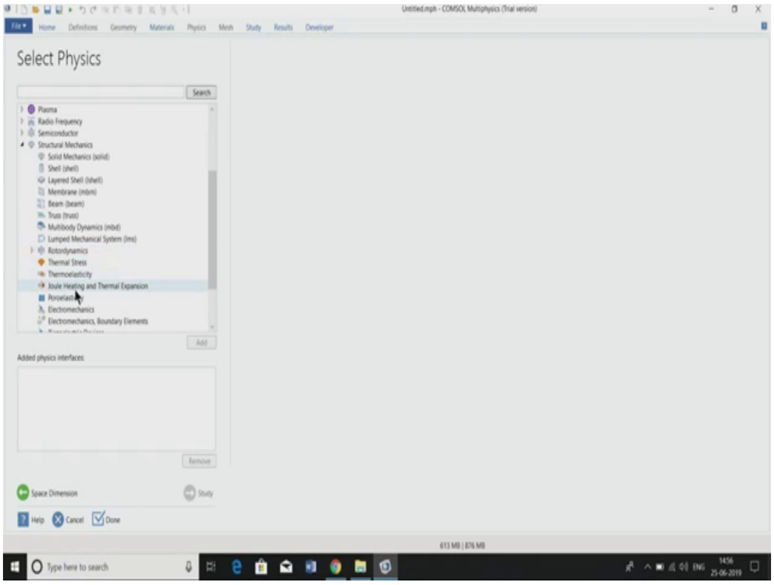
Plasma simulations.

(Refer Slide Time: 04:28)



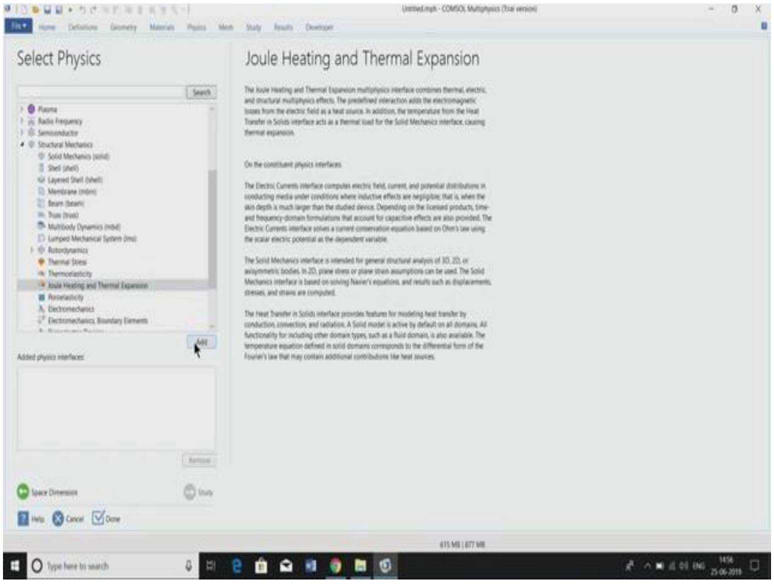
Optic simulations, ray optics, wave optics. So, let us go to structural mechanics.

(Refer Slide Time: 04:38)



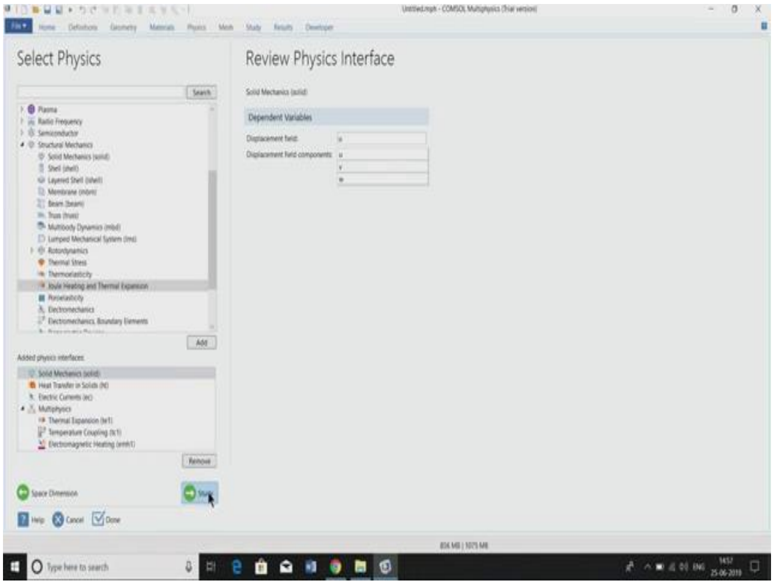
So, we will go to structural mechanics, here we have to go to joule heating because what we will be doing is, joule heating joule heating and thermal expansion.

(Refer Slide Time: 04:46)



Click on it and add it. So, it will take some time to add it.

(Refer Slide Time: 04:56)

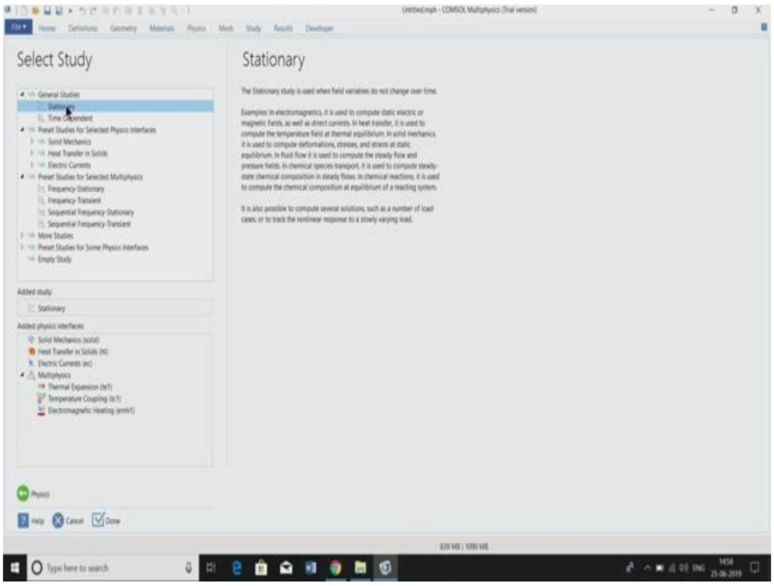


So, it has added the joule heating and thermal expansion physics here, the added physics you will be able to see here, ok; solid mechanics, heat transfer in solids, electric current is there, multi physics what are multi physics are there, thermal expansion is there, temperature coupling is there, electromagnetic heating is there mechanical deformation is there.

So, the idea is that when there is a structure and when you heated that heating causes the structures mechanical shape to change, ok. And another thing is when you have a structure which is metallic you pass current through it because of the non-idealities in the structure there will be heating, it is the I square are heating. When you pass current through a resistor, the resistor will get heated up right, that is called I square are heating. Because of that I square are heating, the structure itself will get heated up; when the structure gets heated up it will affect the mechanical properties. So, this is how different physics it will talks to each other and affects each other. So, this multi physics simulation is what we are trying to see here.

So, now we have added the physics from the structural mechanics module here, we have gone there, we have gone to joule heating and thermal expansion and we have added the physics here, now we will go to study, this is the study.

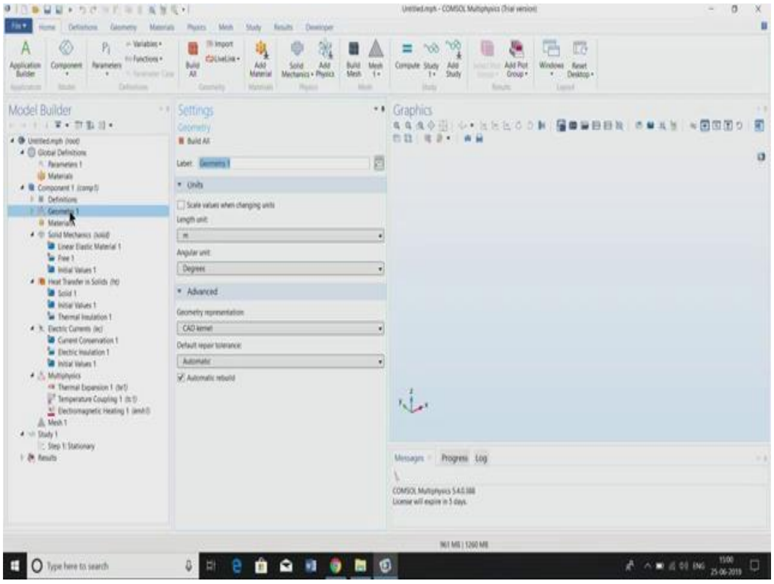
(Refer Slide Time: 06:15)



So, for the time being there will lot of study that you can perform, we can perform stationary study, time dependent study and pre search studies are there all those things. So, today what we are going to do is, we will do a perform stationary study. What is the stationary study? A stationary study is that, the moment you apply a change in a system you wait for the system to respond to that change and stabilize, then we see what is the state of the system at the stable state, that is the stationary study. What is a time dependent study is, as and when you apply a change to a system, there will be a transient response and a steady state response. A transient response it is a immediate response of the system which dies down over time and the study state response is the one that is a stable non-changing response of that imbibes the changes that have occurred to the system.

So, we are going to study stationary; do you are going to a stationary study at time dependent study will involve both the transient and the steady state both together. Let us do a stationary study and then I have to double clicked it, here waiting for it respond and done.

(Refer Slide Time: 07:29)

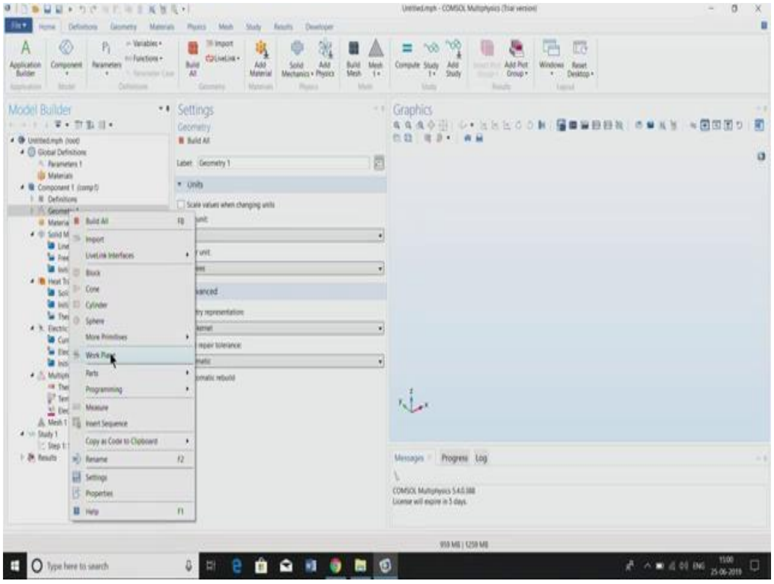


Now, this is the pain of the structure, this is there all the simulation everything we will be doing.

So, now if you look at the left side you will have components; what is the main component? In the component there are main things one is the structure how do we define the structure that comes in the geometry. Next is what are the different physics that are involved? One is heat transfer in solids, electric currents, multi physics solid mechanics. So, it involves three physics solid mechanics, heat transfer in solids and electric currents and then what are the multi physics variables that are there.

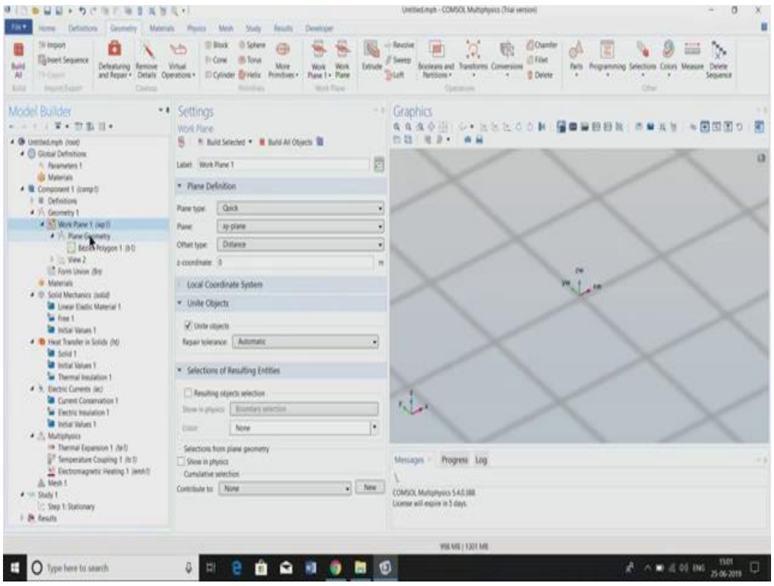
Before we do all those physical studies, we have to first see the geometry right we have to make the geometry. So, in the geometry we have told that we are going to work on a 3D geometry correct.

(Refer Slide Time: 08:37)



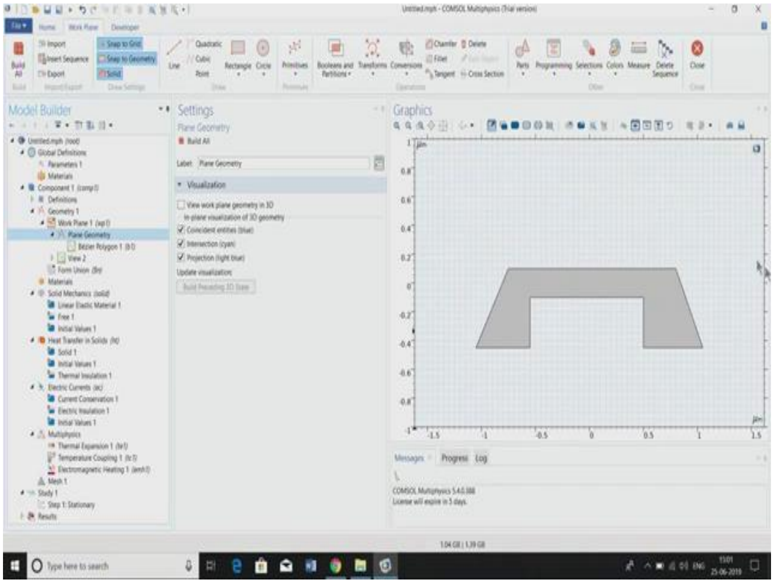
So, left click on the geometry and give work plane.

(Refer Slide Time: 08:40)



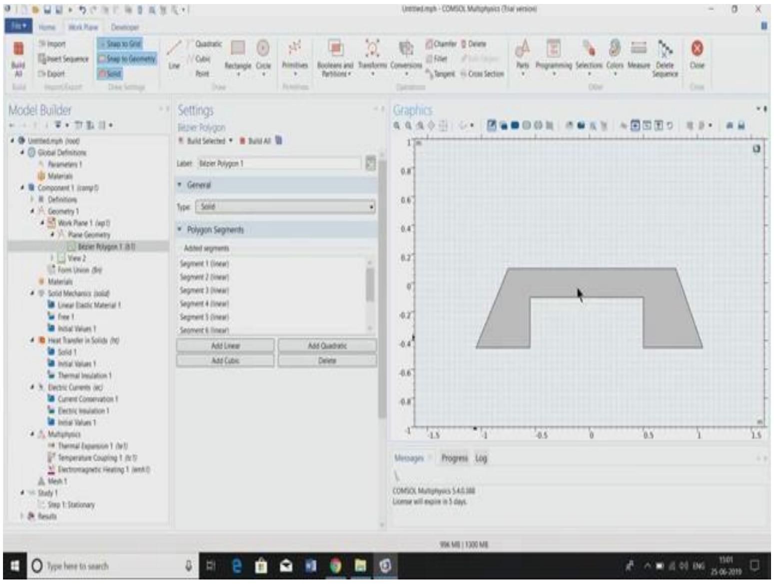
So, this is will be a 2D work plane, ok.

(Refer Slide Time: 08:50)



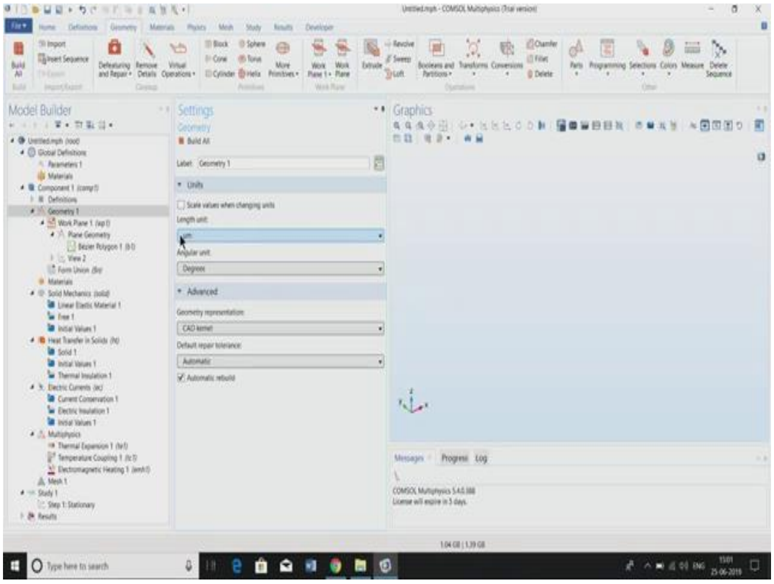
In the work plane, this is a work plane and in that work plane you have a plane geometry which is a 2D space ok. Once we are in; this is the 2D space of the geometry, once we are in the 2D space we can actually draw a structure I am going to draw a structure how do we draw the structure? You go here up work plane and select line here. Once you select line, you can make the structure here that is what I am trying to do.

(Refer Slide Time: 09:40)



So, I have made the 2D structure we can see this correct.

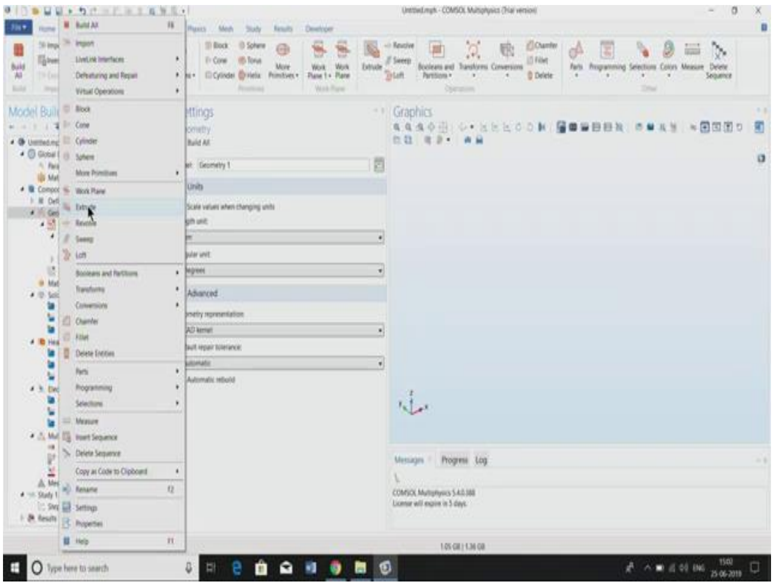
(Refer Slide Time: 09:53)



Now, go to the work plane, you go to the plane geometry, you have the plane geometry go to geometry now you are in geometry. So, most of the times we especially in MEMS devices we worked in micrometer length scales. So, give the length scale here as micro meter it is micrometre length scale and you have the work plane correct.

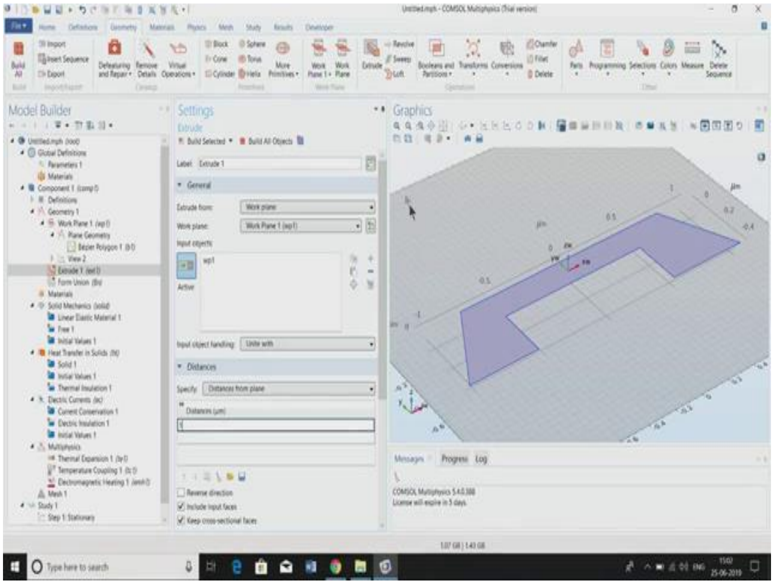
Now, what you have to do is, this is the 2D geometry correct, we need a 3D geometry. So, what we do is, we extruded what we call it is we push it pull it up in the required dimension. So, that is called extrusion.

(Refer Slide Time: 10:27)



So, let us extruded; so, you have to click extrude here, right click on geometry and click extrude.

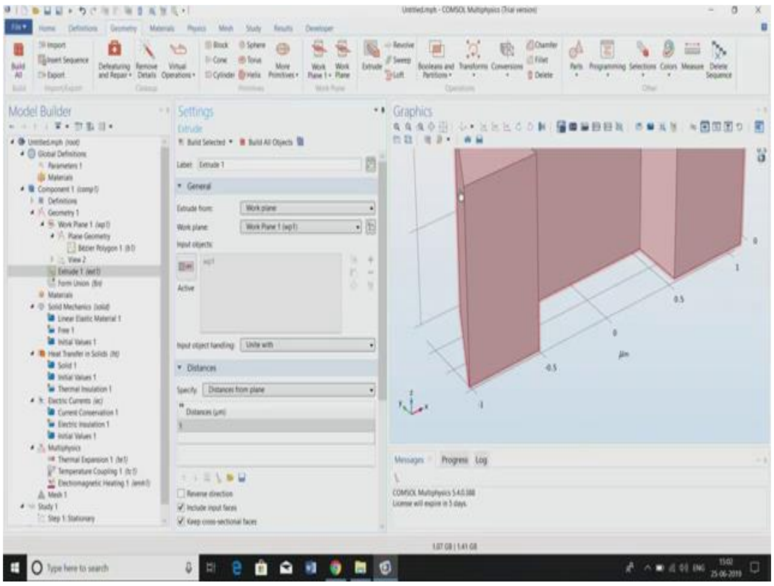
(Refer Slide Time: 10:32)



When you tell extrude, how much distance do you want extrude that is given here. So, I have given 1 micro meter extrusion, let us keep its 1 micrometer, this is the distance to be extruded you can see that.

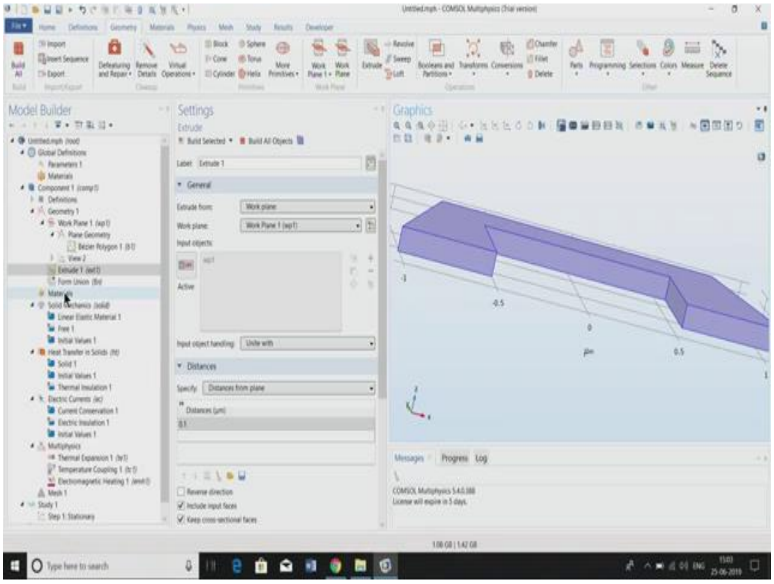
Now, once we have given one, you can see that right now it is only 2D still in the 2D, correct.

(Refer Slide Time: 10:59)



Now, we will give build all objects when we give build all objects it will become a 3D so, but this is too big as you can see. So, let us make it 0.1 micrometer and again give build all objects.

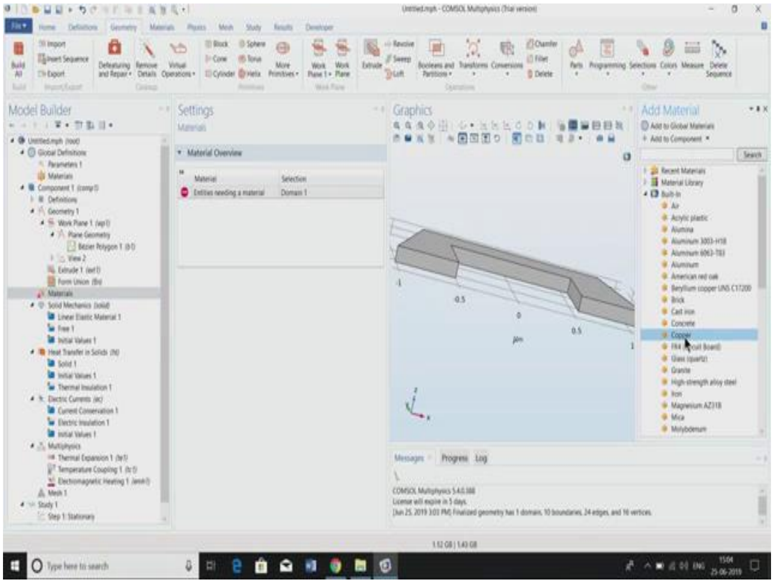
(Refer Slide Time: 11:10)



So, you will get a 3D geometry now, you can see right there is a 3D geometry. So, this way your geometry is ready, we can see the geometry that has been formed I am rotating and showing you. It is a micro system beam that beam can be mounted like this on a pcb or something its very small that can pay currents.

Now, the geometry is ready, now next thing is as I told you the first thing that happens is the flow of electric current. Before electric current can flow we need to give some physical meaning to this thing right. What is the biggest physical meaning to any structure is the, what kind of material it is made up of. So, first thing that you have to go is material this material is here. So, main things component, geometry, material and the physics these are the main things.

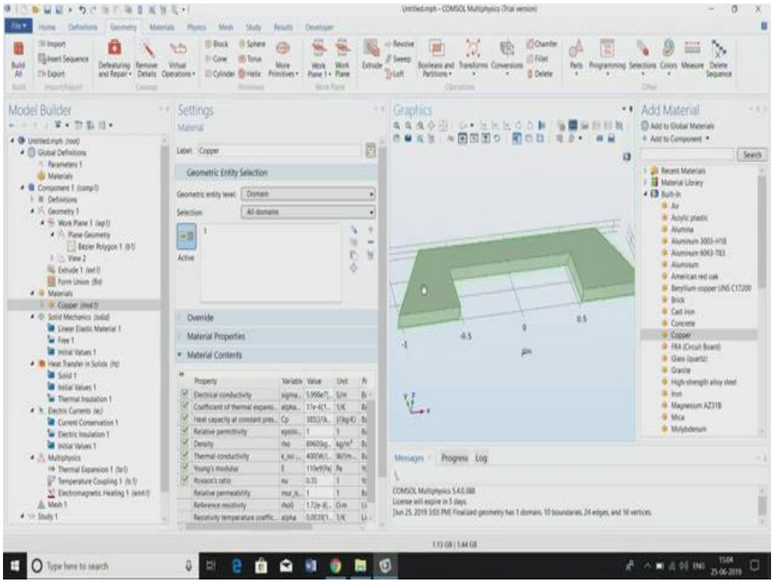
(Refer Slide Time: 12:08)



Go to the material right click, click add material from library, ok.

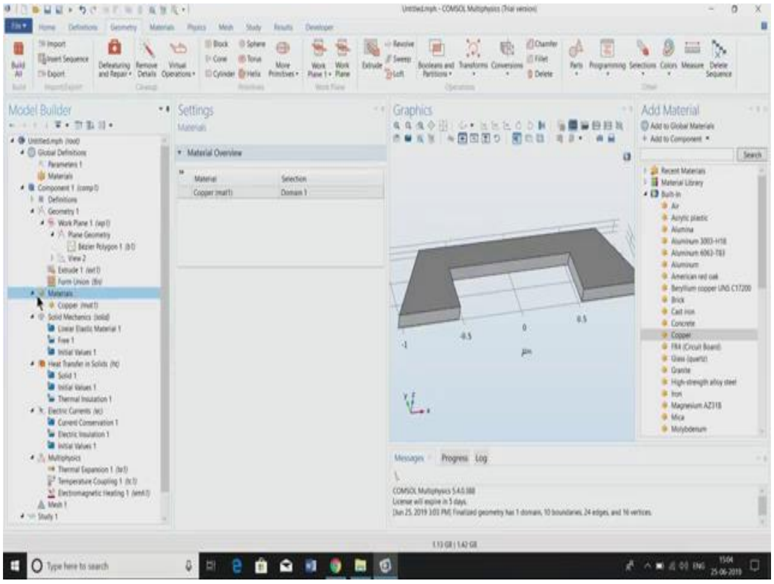
So, in here you can go to built in materials, click it will load and then we give copper.

(Refer Slide Time: 12:27)



So, now all domains; so, each of these interface is called the domain each of this faces are called a domain all the domains have been assigned the material as copper. So, material selection is done.

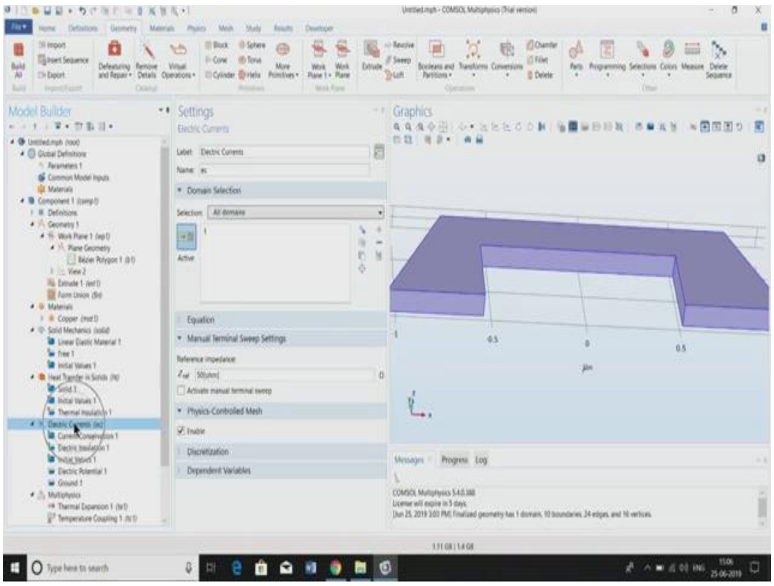
(Refer Slide Time: 12:45)



Once material selection is proper, if a solid mechanics is not done correctly, here you will see an into mark now that a into mark is not there.

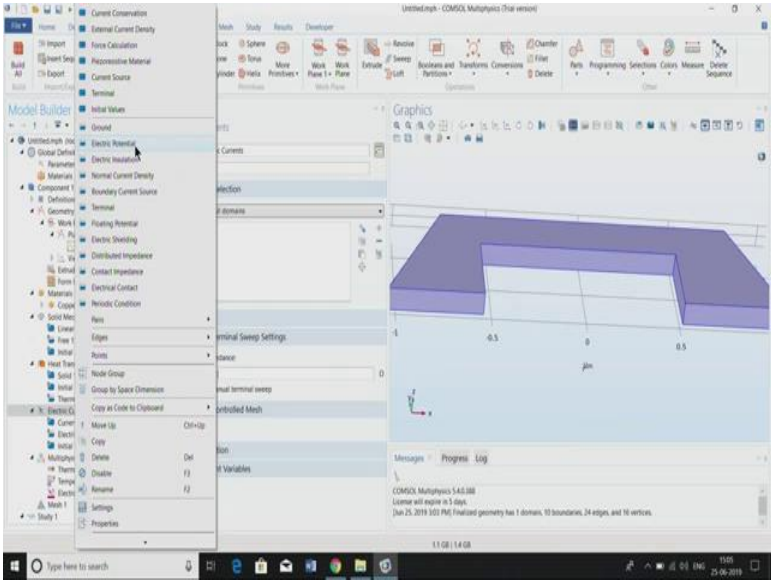
Next is now that your structure is ready, your material is defined, now next what do you do? As I have explain to you the first thing that happens is the flow of electric current, flow of electric current causes heating I^2R heating, the heating causes changes in mechanical properties.

(Refer Slide Time: 13:19)



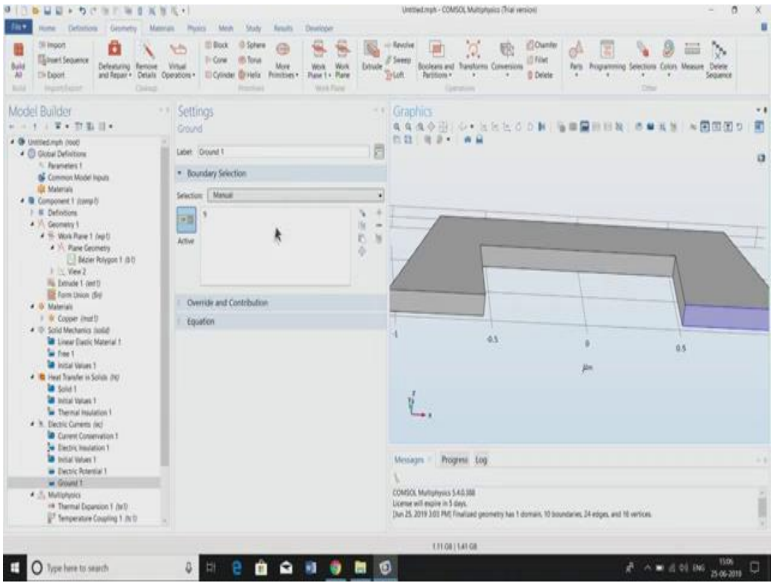
So, first thing that we have to do is go to the electric current module here right click, if it pass an integral what do you need? You need a point where you inject the voltage and you need the point where the voltage the current will go out that is the ground.

(Refer Slide Time: 13:29)



So, right click and put electric potential.

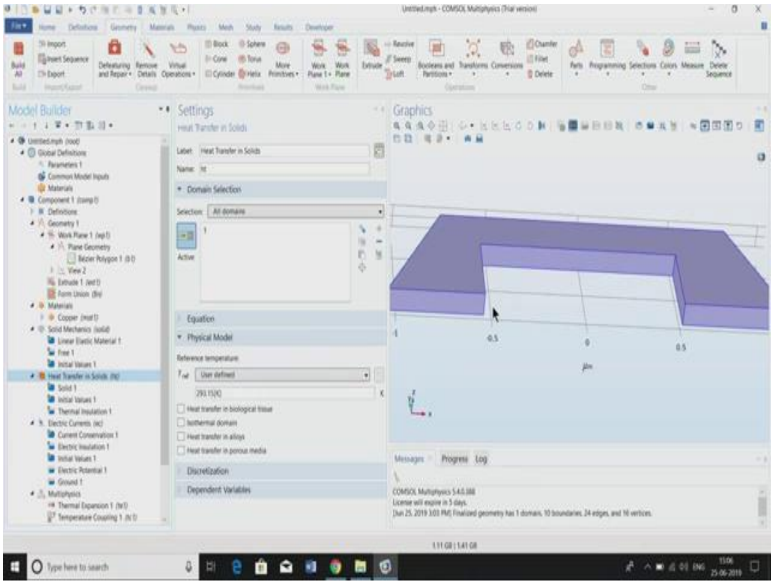
(Refer Slide Time: 13:31)



So, your electric potential comes you can give the electric potential value as 0.5 volt and giving the value here, ok.

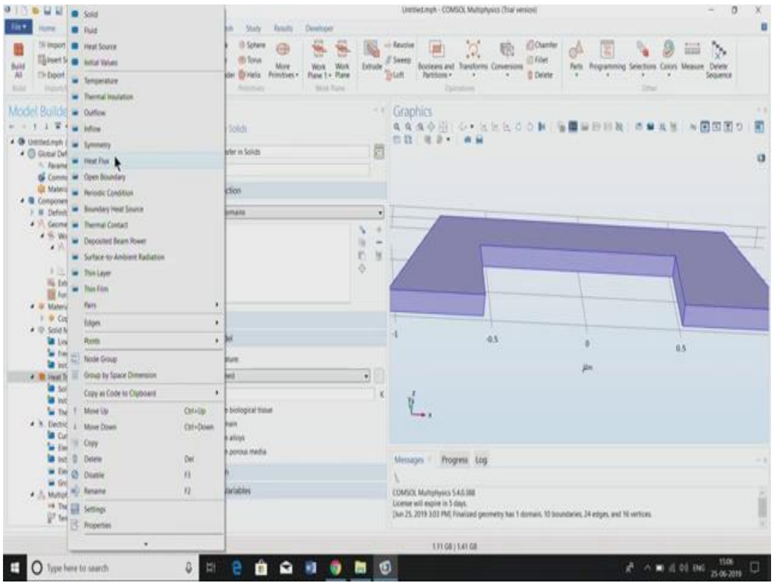
Now, next thing is we have to assign where it will be. So, as this let us give electric the region coming in red colour, let us give this region the voltage. So, I am clicking it. So, as soon as a click it that number come here, see if I click here it will come here, but I cannot give two places, right. So, I will delete this. So, this is our injection point of the voltage correct next thing what we need is ground. So, ground is also assign here. So, that is done electric current is done next is heat transfer it will get heated right.

(Refer Slide Time: 14:30)



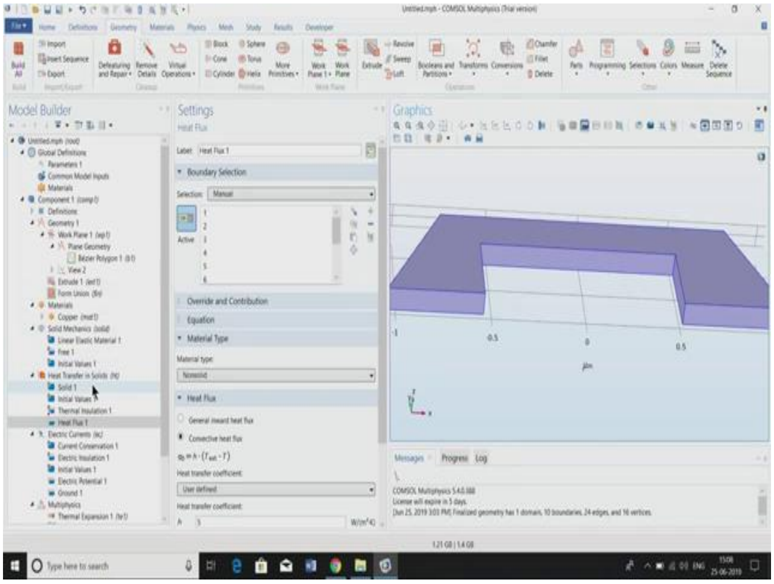
So, go to heat transfer, now when it gets heated when you are doing a multi physics simulation you need a give boundary conditions. As an object cannot get heated in isolation correct, it will get heated and that heat gets transferred by the surrounding that it is in; sometimes it will be air, sometimes it will be water cold, it can be anything conserving it is an electronic system as a MEMS device most of the time it will just air, correct.

(Refer Slide Time: 14:59)



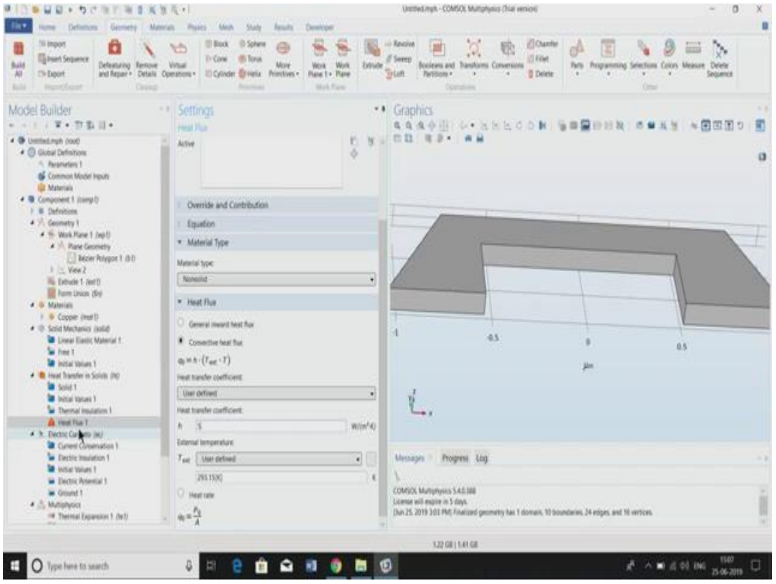
So, we have to give that constraint. So, that heat flux will be there which is the flow of heat. So, go left click on heat transfer and click heat flux here, heat flux is there.

(Refer Slide Time: 15:11)



Now, what is the heat flux? Hours will be a convective heat flux because it will be air cooled. So, click convective heat flux and what is heat transfer coefficient user defined, heat transfer coefficient usually for air is 5. So, we give 5 here, I have given the heat transfer coefficient here as you can see.

(Refer Slide Time: 15:33)



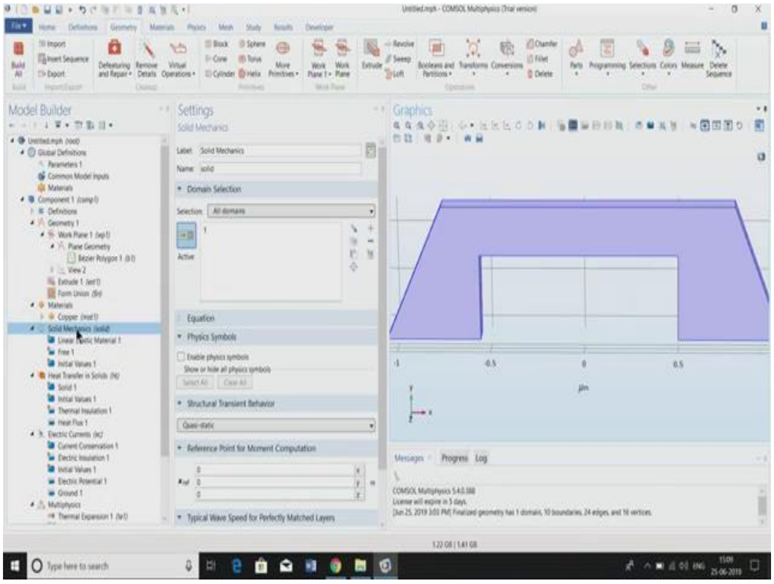
And we have to assign the whole area. So, still even now this is showing an alarm signal; that means, there is something wrong in how we have define this. See the softwares can be used only. So, it is only as intelligent as the person using the software, if you do not know exactly what you are looking at what no kind of software can help you.

So, in this course we are showing you this software there are lot of other softwares ANSYS, CATIA lot of softwares are there we showing this to you as an example ok. So, just because we are using this so, we are familiar with this. So, we thought, but the idea is to show you how any software can be used to simulate MEMS devices correct. But then if you do not know the physics that is involved exactly, then you not be able to get the expected result out of it.

So, you should understand your problem very thoroughly what are the conditions, where you will injured voltage, what will be the dimension of the structure, what will be the different physics that will be involved once you understand everything then only you can simulate, you cannot do a blind simulation ok. So, now, we; so, this thing is showing and I am write, because we are not assign the boundary for it; I mean, what where it will be applied. So, just go inside this work plane control a, all everything got assign for the heat flux. So, that is done.

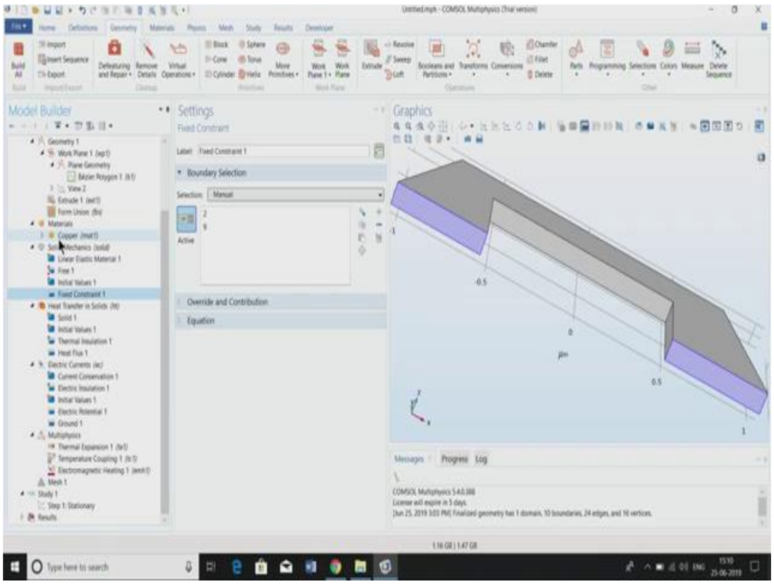
Now, heat transfer is assigned electric current is assign. Now, we have to see solid mechanics.

(Refer Slide Time: 17:09)



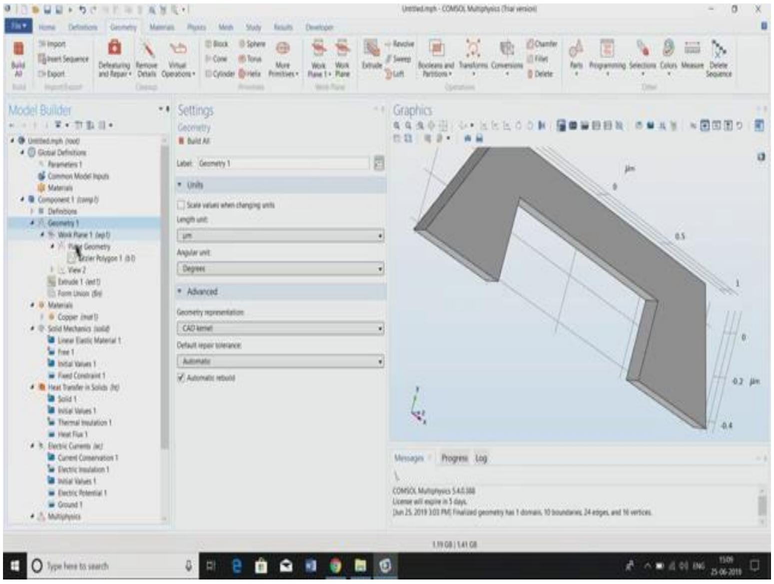
Now in solid mechanics the idea is that, for an objective bend somewhere it has to be fixed somewhere right, if it is not fixed anywhere it cannot bend. So, this is a micro resister beam just like any other beam, which will be fixed in this place and this place on some pcb or somewhere. So, these two are the points which be fixed so, that will be a constraint. So, right click on solid mechanics and click fixed constraints here.

(Refer Slide Time: 17:39)



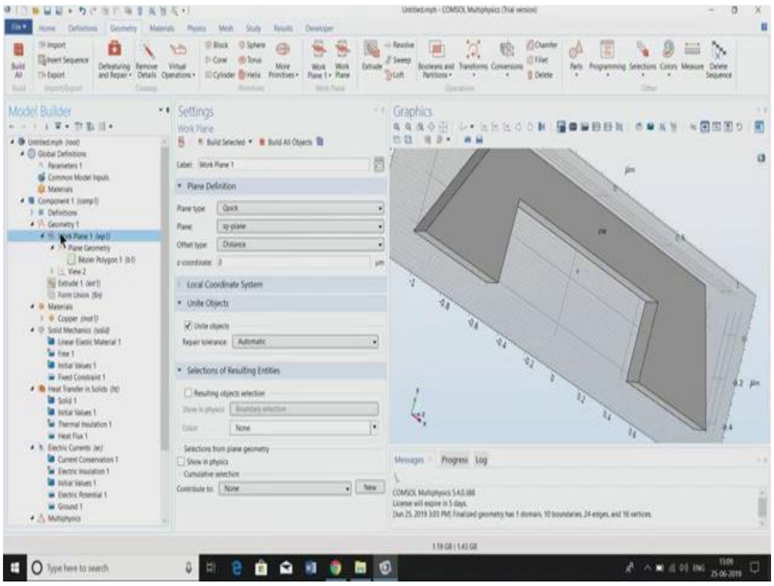
When you click fix constraints, you have to assign which are the fix constraints. So, just click here and here, these are the fixed constraints; you understood, right. So, we are assign the fixed constraints, correct.

(Refer Slide Time: 18:03)



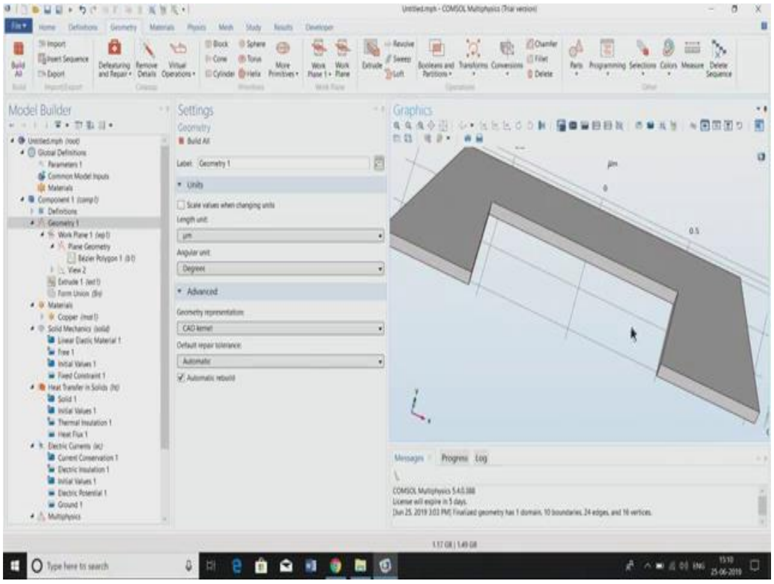
So, now what are we have done? We have gone to the geometry correct, we have made a plane geometry then we have extruded it.

(Refer Slide Time: 18:14)



We got the 3D model of it in the work plane.

(Refer Slide Time: 18:16)



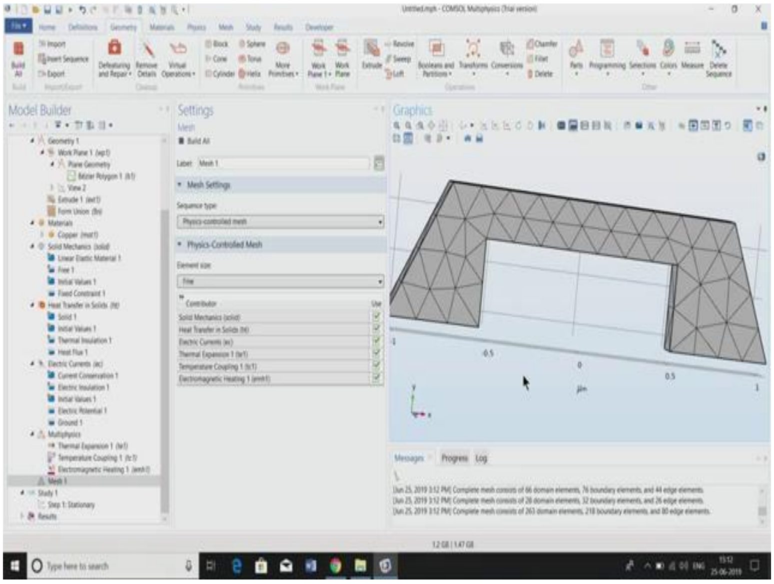
The geometry is defined. So, you can see the geometry ok. Then what we did? We fix the electric current where actually be applied, what will be the voltage that you will give everything you have fixed like here, where is the electric potential, where is the ground all those things. Before that we fix the material what kind of the material it is, which domains it will be applied, supplied everywhere and the material is copper.

Then you fix the electric current, then we went to heat transfer we told at the heat transfer of well occur as a convective cooling through air, then we gave the heat transfer coefficient through the heat flux correct. Then we went to solid mechanics, there will be some constraints that will be fixed. So, we gave the fix constraints, what are the fix constraints we gave here, so, all these things are done.

Now, the next thing is what thus this or any other simulation software do. What it does is it will know have an engine, where all these physics or modern as equations we given the constraints that we have given, it will solve for these equations at every point defined on the structure.

Now, the prediction of the solution will be more accurate, if we have more points to solve for these equations correct; so, but then it will involve more computational resources. So, that is where this concept of meshing comes.

(Refer Slide Time: 19:54)

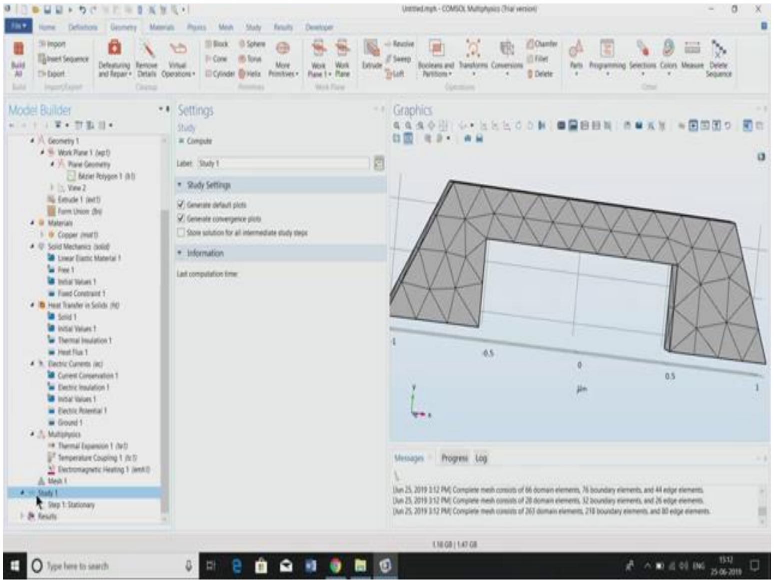


We can see here this is the mesh. Click on the mesh, physics controlled mesh is one option or use a defined mesh, physics controlled mesh is when it will know the minimum amount of machine that is required to give a solvable solution not a above the value that is bounded out that is a solution that is bounded.

So, we have given physics controlled mesh and they, we will build it. So, when we build it you can see that meshes right; wherever these meshes are conducting all along these lines questions will be solved now if you want a more accurate solution, we can give a higher meshing. So, I let me see if you can see lot of meshing options are there here extremely course, extra course, course or course normal fine. So, let me give finer and then if I give build all you will see a more dense meshing, correct. Now if I give courser will be thinner meshing.

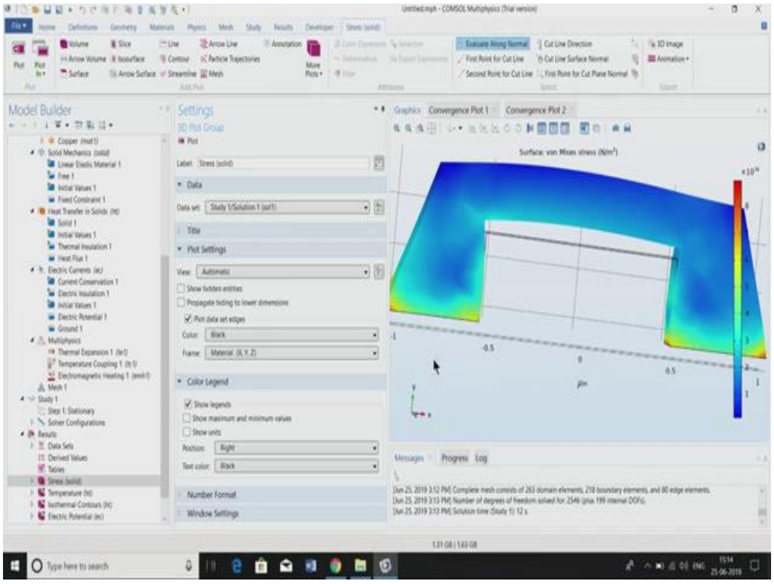
So, if you give very; extremely course meshing, the system the software will not able to solve because it does not have sufficient points to solve for the entire structure. So, let us give a fine mesh. So, this will give a very smooth results ok. Now we have given the meshing we have define the physics involved, we have made the structure now the only thing left is to do perform the study.

(Refer Slide Time: 21:18)



So, far that you go to study and there is a default studies that are available we have going to study, then we are going to compute here it will compute the results. So, just observe here that this interface is here is right angled in color as shape, here also these are all perfect lines, ok. Once we compute we will see the effect of mechanical deformation on this I am computing, click compute here. So, it is computing the solution, it will take some time depending on the CPU and RAM of your meshing.

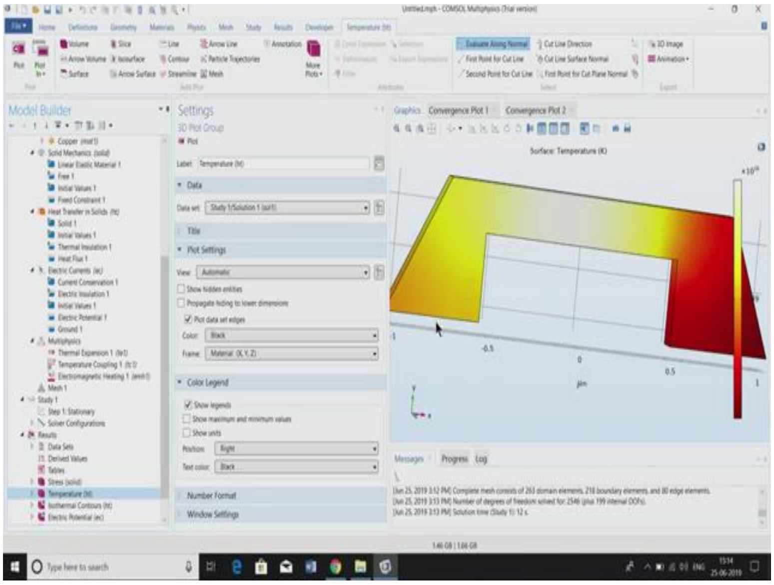
(Refer Slide Time: 21:59)



So, it has computed, we have got the result you can see here right now see it has bent, the structure has bent. This is the mechanical deformation that I was talking about here also it has bulged out. If you give more voltage, it will get heated up more and it will bulge more that also we can see soon.

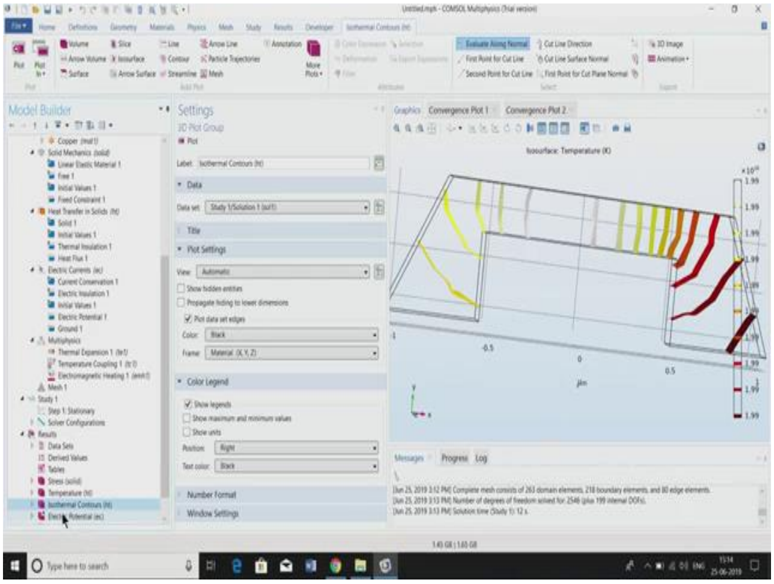
Now, what is this plot? So, this study is the study about the stress where are the stresses high. The stresses are maximum at the fix places where we are fixed it of it is very intuitive.

(Refer Slide Time: 22:42)



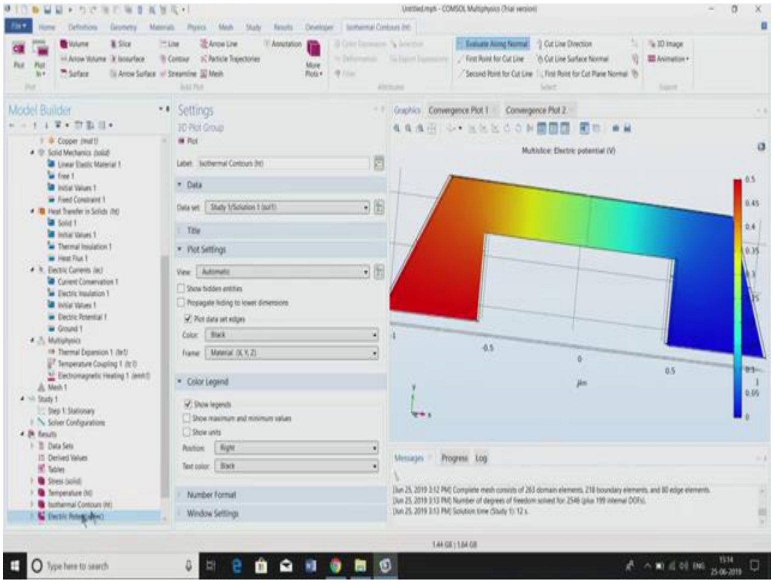
Now, let us see the temperature plot here, click temperature you can see how temperature is varying. As this expected where you give the voltage there is the temperature will be high and the temperature is maximum here at the junction and it cools down

(Refer Slide Time: 22:55)



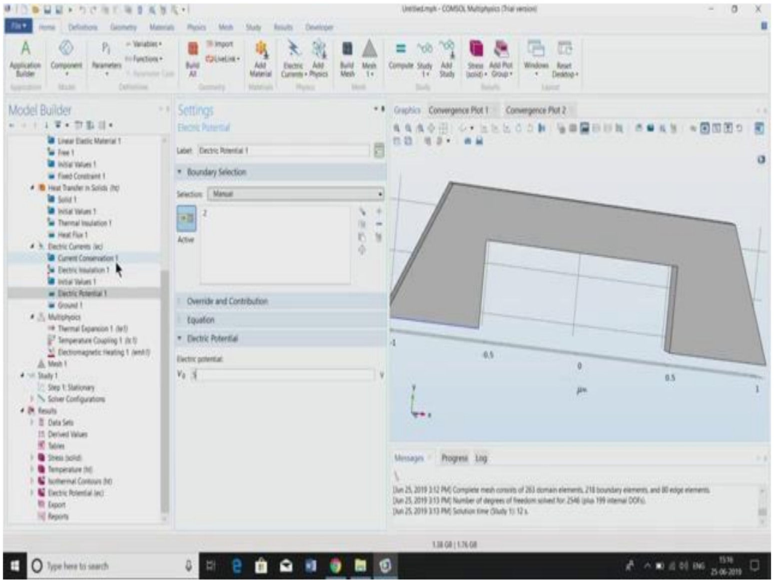
Next we can see isothermal contours, where all there is uniform. So, isothermal contours are there at the direction perpendicular to the flow of electrons are isothermal contours.

(Refer Slide Time: 23:04)



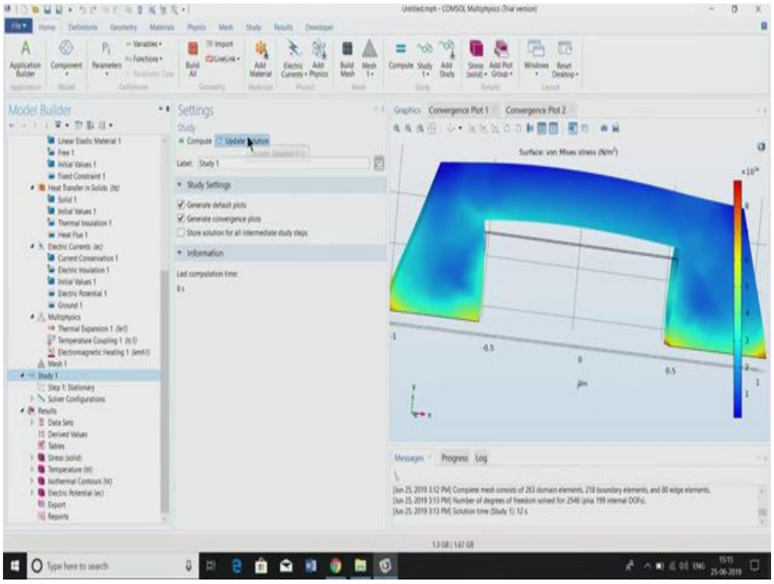
Then how is the electric potential changing we can see. It starts of at 0.5 here, that is here and it ends at 0 volt here, ok. So, these are the basic studies at you can perform you can see the stress changing, temperature changing isothermal contours changing and all.

(Refer Slide Time: 23:27)



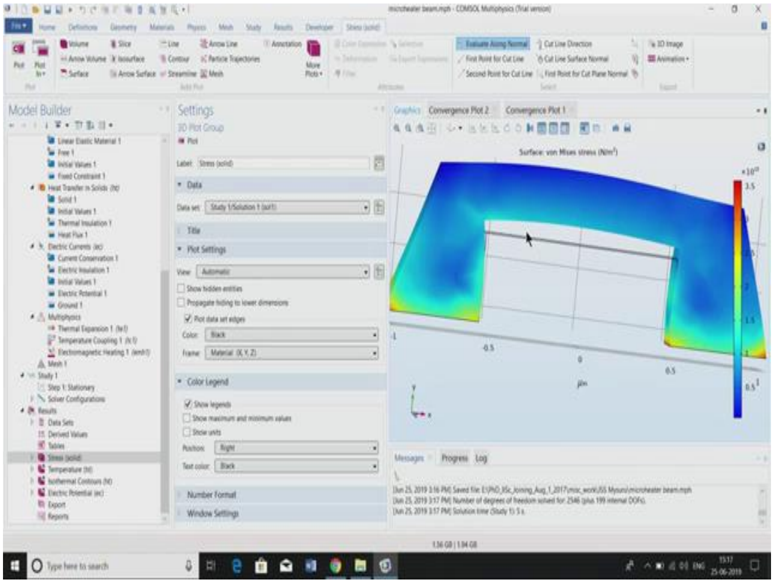
Now, let us do a different thing let us change the electric potential, I am working at one volt we will see how the deformation changes. I have made it one volt go to geometry again do a build all, we go to meshing we have given fine meshing now we will go to study and let us update the solution.

(Refer Slide Time: 23:56)



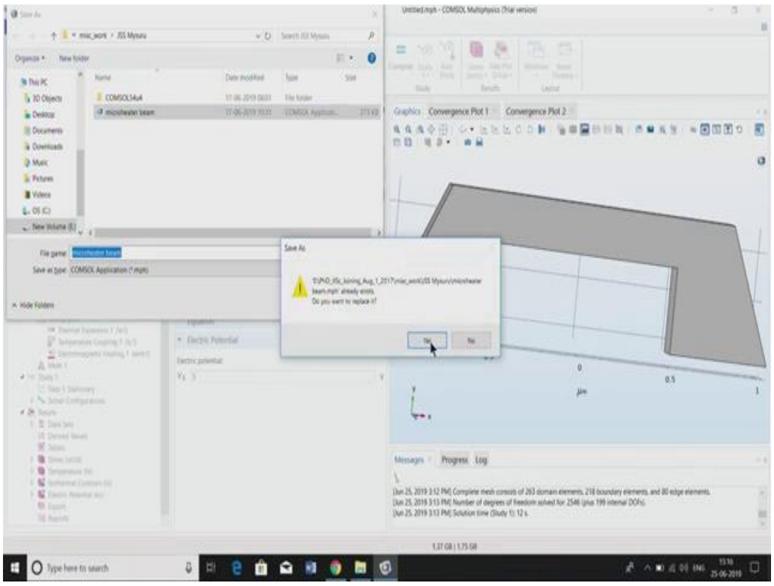
I will go to the stress plot, this has the original stress plot correct, I am going to update the solution here, you will get the option updating solution. So, it is updating the solution.

(Refer Slide Time: 24:04)



So, the bulging has increased here if you can see, see here. So, let me try to change it further let me give 5 volt let say.

(Refer Slide Time: 24:30)



I am just saving it as micro heater beam. Now the electric potential is 5 volt, let me go to the study and I will let me give compute again. You can do compute or update solution anything is fine.

See we can see that there are minute changes the bulging has increased. Let us go to a lower voltage, let us say I give 0.1 volt. Let me build it all go to study again and then compute. So, you can see the bending is not that much like before especially this vertical beam bending. So, like this you can engineer your design, before actually making it to see that what should be the thickness of my beam, what should be the material that I should use, what should be the voltage that I should give, what should be the type of cooling that should I give all these design constraints for your sensors or actuators can be very well studied using such simulations before you actually make the device.

So, in this is simulation is very important provided you understand the basic concepts that you want to work with it. So, if you are engineering something there will be few parameters that are in your hands right, few design parameters like the dimensions, the voltage that you apply. So, how do you arrive at a very good combination of such design parameters that you can do by doing such simulations. So, that is the importance of such simulations.

In further lectures in between other lectures we will keep showing you other interesting device simulations. Today we saw the simulation of a micro heater beam micro resister beam or micro heater beam, which showed you the coupling effect of three different physical properties of a material like electrical, thermal and mechanical properties correct, and seen how applying a electrical voltage changes the temperature across the material and also causes mechanical defamation. So, such physical studies can be performed with such software. So, this is overall idea of showing these things to you.

Thank you and see you in the next module.